CONGRATULATIONS!

You have just purchased MILL*PWR*® by ACU-RITE®, a versatile and flexible 2- or 3-axes Control/3-axes Readout System that effectively combines powerful features and functionality with ease of use… at an affordable price.

MILL*PWR* was developed to satisfy the wants and needs of tool and die makers and other machinists where manual and automated operation are both useful and needed. MILL*PWR* will enable you to maximize your throughput by significantly reducing set-up time, scrap and other non-productive operations thereby increasing your efficiency, productivity and profitability.

MILL*PWR* was engineered and is manufactured in the United States at ACU-RITE's ISO 9001 registered facility. MILL*PWR* is a complete system that includes ground and hardened ball screws, powerful servo motors and a user-friendly operator console that includes a 10" flat-panel, color, LCD display and a built-in floppy disk drive.

MILL*PWR* is a closed-looped system with positioning feedback provided by ACU-RITE precision glass scales (2µm/0.0001" resolution). MILL*PWR* also includes Position-Trac™, an advanced, unique feature that enables you to easily, quickly and accurately re-establish workpiece zero after power loss.

MILL*PWR* utilizes a conversational, menu-prompted format that makes it easy for you to learn and quick for you to program. No prior programming experience or training is necessary. All you have to do is simply enter part dimensions directly from the print. MILL*PWR* automatically calculates the tool path... with immediate-part-view graphic feedback providing program verification. MILL*PWR*'s intuitiveness allows you to learn how to operate MILL*PWR* and begin making parts, and profits, in a matter of hours.

MILL*PWR* is backed by a comprehensive one-year warranty, with nationwide support provided by a factory trained and certified distribution network.

Thank you for choosing ACU-RITE. We're confident you'll be more than glad that you did.

Sincerely,

ACU-RITE

MILL*PWR* Setup Access Code

An access code must be entered before the installation setup parameters can be accessed or changed.

IMPORTANT
The access code is **8891**.

Refer to Section 7, **Setup**.

IMPORTANT
Supervisors may wish to remove this page from the **MILL*PWR*** Operation Manual after initially setting up the installation setup parameters. Keep it in a safe place for future use.

Introduction **1-1**
System Overview 1-1
Operator Console Overview 1-2
Keypad Layout 1-3
Screen Layout 1-4
Table Stop Button 1-5
Conventions 1-6
Axis Conventions 1-6
Count Direction 1-6
Cartesian Coordinates 1-7
Polar Coordinates 1-7
Absolute and Incremental Dimensions 1-8
Z-axis Conventions 1-9
Z-axis Retract 1-9
Begin and End Depths 1-9
Pass 1-9
Disengaging the Z-axis Drive System 1-10
Setting Z-axis Datum when Changing Tools 1-12
Drilling Conventions 1-12
Peck 1-12
Tool Retract 1-12
Dwell 1-12
Saving, Backing Up, and Creating Directories 1-13

DRO **2-1**
Start Up 2-1
Power Up 2-1
Screen Saver 2-1
Finding Home 2-2
DRO Functions 2-4
Move Table 2-4
Zeroing an Axis 2-5
Inch/millimeter 2-5
Teach Position 2-5
Using an Electronic Edge Finder 2-6
Skewing a Part 2-7
Establishing Datum 2-9
Hard Key Milling Functions 2-12

Programming **3-1**
Programming Considerations 3-1
"From" and "To" Points 3-1
Depth of Cut 3-1
Pass 3-1
Tool Offset 3-2
Datum Selection 3-2
Absolute and Incremental Dimensions 3-3
Continuous Milling 3-3
Creating a Program 3-5
The View Key 3-7
Follow Tool 3-7
Show Tool Path 3-7
Zoom In, Zoom Out and Restore 3-7
Running a Program 3-8
Skewing a Part 3-8
Establishing Datum 3-10
Testing Your **MILL*PWR*** Program 3-13
Single Step 3-13
Dry Run 3-13
Graphics Only 3-13
Manual Positioning 3-14
Disable Look Ahead 3-14
Machining Your Part 3-15
Feed + and Feed - 3-16
Manually Positioning the Quill 3-16
Program Functions 3-17
Directories 3-18
Creating a Sub-directory 3-19
Selecting a Directory 3-21
Deleting a Directory 3-23
Saving a Program 3-24
Naming a Program 3-25
Loading a **MILL*PWR*** (MPT) Program 3-26
From **MILL*PWR***'s internal memory 3-26
From a 3½" floppy disk 3-27
From your PC 3-28
Translating a DXF file 3-28

Loading a G-code file 3-29
Running a G-code Program 3-31
MILL*PWR* - G-code Conventions 3-33
Merging Programs 3-37
Backing Up a Program 3-38
Deleting a Program 3-39

Demonstration Program 4-1
Selecting Datum 4-1
Beginning the Program 4-2
Selecting a Tool 4-2
Programming the Contour 4-4
Programming the Bolthole Pattern 4-12
Programming the Rectangular Pocket 4-15
Saving Your Program 4-17
Testing Your Program 4-18
Running the Program 4-19
Tool Changes 4-20
Clearing the Program 4-20

Program Steps 5-1
Simple Milling and Drilling 5-1
Programming a Tool Step w/Repeatable Tool Length Offsets 5-2
Entering the First Tool 5-2
Changing to a Tool of Unknown Length in the DRO 5-4
Changing to a Tool Of Unknown Length in PGM 5-4
Position/Drill 5-6
Centerline 5-7
Line 5-8
Arc 5-9
Blend 5-11
Rectangular Milling Functions 5-13
Pocket 5-13
Frame 5-16
Face 5-18
Slot 5-20

Circular Milling Functions 5-22
Pocket 5-22
Frame 5-24
Ring 5-26
Helix 5-28
Hole Patterns 5-30
Row of Holes 5-30
Hole Frame and Hole Array 5-32
Bolthole Circle Patterns 5-35
Additional Milling Functions 5-37
Custom Pocket 5-37
Repeat 5-39
Rotate 5-40
Mirror 5-41
Contour 5-42
Engrave 5-44
Engrave Line 5-44
Engrave Arc 5-47
Ellipse Frame 5-49
Chamfer 5-51
Reference Point 5-53
Island 5-54
Spiral 5-55
Comment Step 5-57
Auxiliary Function (AMI Option) 5-58
Step Functions Softkey 5-60
Explode 5-60
Reverse Step 5-62
Reverse Path 5-62
Change Steps 5-63
Delete Steps 5-64
Copy/Move Steps 5-65

Calculator 6-1
Math Functions (+, -, ÷, x) 6-2
Trig Functions 6-2
Geometry Functions 6-4

Working with the Geometry Calculator 6-4
GeoCalc™ Quick Reference Table 6-5
Using GeoCalc 6-6
Calculator Functions 6-7
Saving Results Calculated in GeoCalc 6-7
Loading GeoCalc Results That You Previously Saved 6-8
Loading Programs into GeoCalc 6-9
Clearing GeoCalc 6-9
Example Problem 6-10
Strategy 6-10
Starting the Program 6-11
Entering the Lines 6-11
Finding the Arc 6-14
Finding the Points of Tangency 6-15
Returning Features 6-16
RPM Functions 6-18

Setup 7-1
Job Setup 7-2
Tool Library 7-2
Using the Tool Library with Tool length offsets 7-4
Entering the First Tool 7-5
Using the Tool Library 7-7
Scale Factor 7-11
Display Options 7-12
Angles 7-12
INC Display 7-12
Display Resolution 7-12
Point Entry 7-12
“From” Point 7-13
Stepover Entry 7-13
Pass/Peck/Retract 7-13
Electronic Edge Finder 7-14
Job Clock 7-14
Feed Rate Settings 7-16
Step Override % 7-16
Max % 7-16

Min % 7-16
Dry Run Speed 7-16
Default Feed Rate 7-16
Unit/(Min) 7-16
Full Cut Feed Rate % 7-16
Installation Setup 7-17
Protection 7-17
Error Compensation 7-18
Encoder Setup 7-19
Travel Limits 7-21
Error Checking 7-22
Serial Port 7-23
Z-axis Control 7-23
Spindle Control 7-24
Error Log 7-24
Servo Tuning 7-25
Automatically Tuning the Servo in Console Mode 7-26
AMI Outputs 7-28
Diagnostics 7-29
Motor Assembly Monitor 7-30
Signal Test 7-31
Table Stop Test 7-32
Circle Interpolation Test 7-33
Examples of Circle Interpolation Test Results 7-34
Keypad Test 7-35
Display Test 7-36
Disk Utilization 7-37
Set Time and Date 7-38
System Statistics 7-38
Max Servo Speed 7-39

Remote Storage 8-1
Equipment 8-1
Choosing a Serial Cable 8-1

Connecting **MILL*PWR*** to Your PC 8-2
Installing the Remote Storage Program onto Your PC 8-3
For Windows® 95 and 98 8-3
For MS DOS®. 8-3
Setting Up Your COM Port and BAUD Rates 8-4
Activating the Remote Storage Feature in **MILL*PWR*** 8-5
Troubleshooting 8-5

Troubleshooting Guide **9-1**
Introduction 9-1
Using the Table 9-1
Table 9-2
Re-aligning (Tramming) the Spindle 9-15

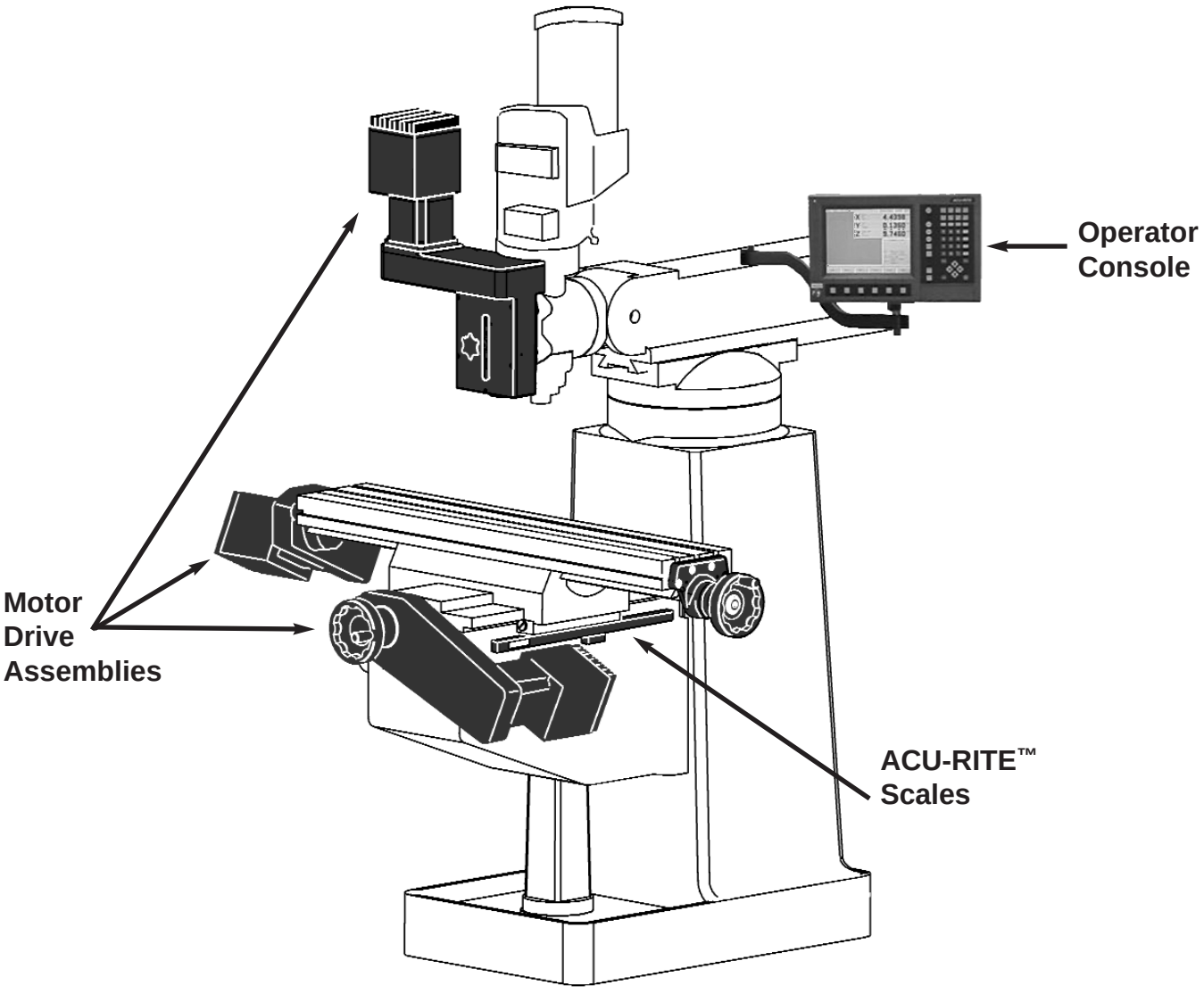
This symbol alerts you to important information concerning the operation of your **MILL*PWR*** system.

MILL*PWR* System Specifications

Characteristic	Specification
Operating conditions	0° to 40° C (32° to 104° F) 25% to 85% relative humidity (non-condensing)
Storage conditions	-20° to 60° C (-4° to 140° F) 25% to 95% relative humidty (non-condensing)
Input requirements: Voltage Frequency Current	 115V~ (±20%), single phase 47 - 63 Hz 8.5A rms nom., 18A rms peak—inrush
Fuse	15A/250V resettable circuit breaker
Encoder input	Position signals, channels A & B TTL square wave signal in quadrature (90° nominal phase relationship) Maximum input rate: 50 kHz Reference signal: TTL square wave
Edge finder input	Compatible with ACU-RITE® Electronic Edge Finder
FCC compliance	Class A

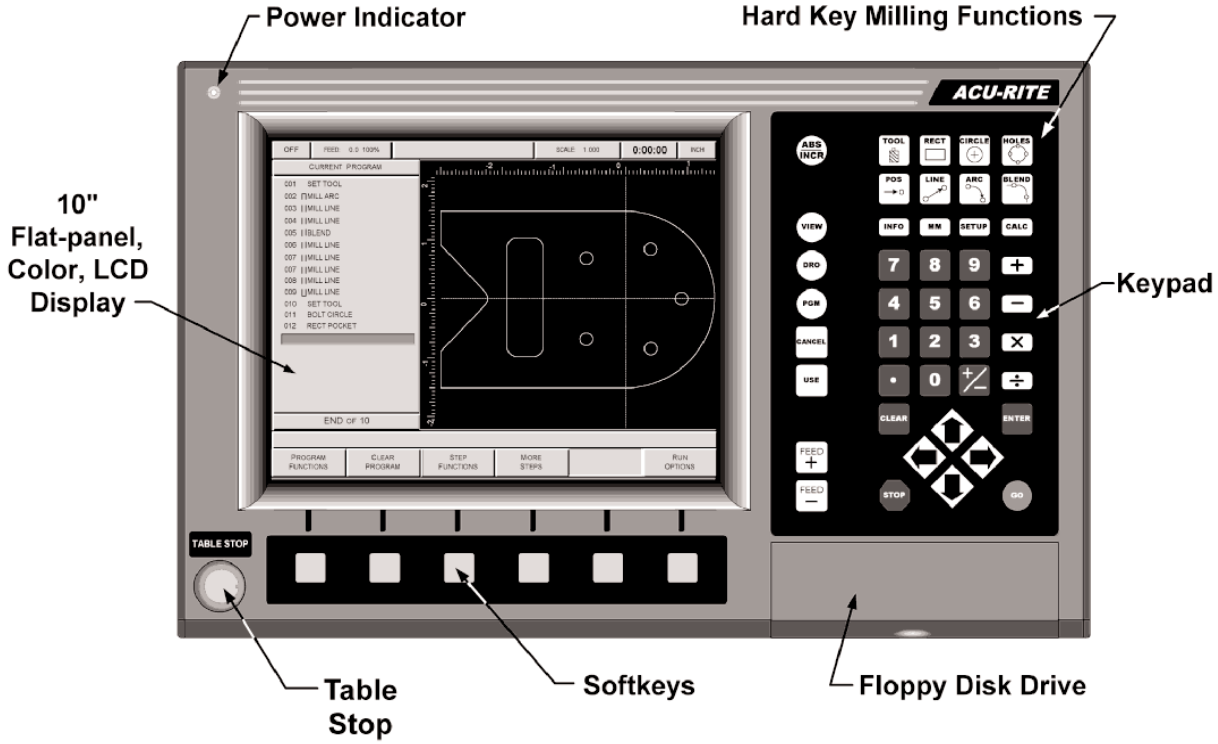
INTRODUCTION

System Overview

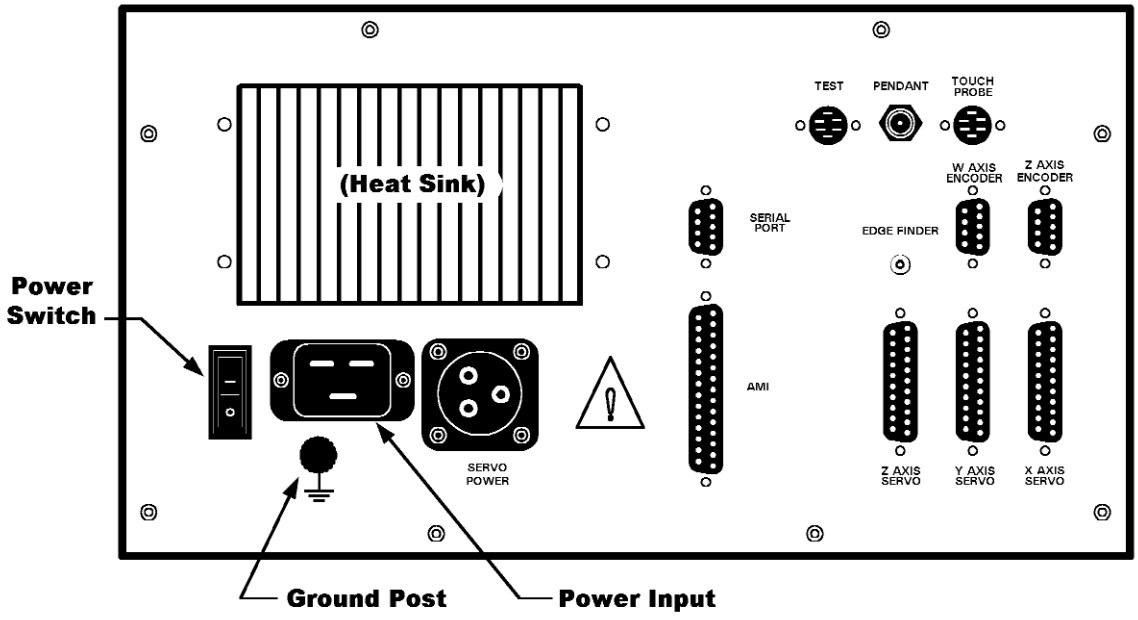


Licensed under U. S. Patent No. 5,941,663
(applies to Linear Encoder systems with Z-axis only)

Front View of Operator Console



Rear View of Operator Console



Keypad Layout

Main Function Keys

ABS INCR — Switch from absolute to incremental (or vice versa) in the DRO display and milling function numeric fields.

VIEW — Manipulate your part graphic.

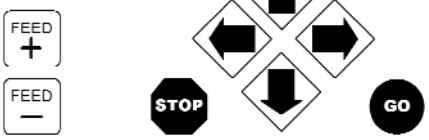
DRO — Display the digital readout.

PGM — Display the program screen.

CANCEL — Exit from a milling function.

USE — Add a step to your program once you've completed an entry form.

Cursor and Motion Control



GO — Start your program.

STOP — Press this key once to pause your program, twice to exit.

FEED+ FEED- — Increase or decrease your feed rate.

ARROWS — These keys enable you to position your table or move your cursor.

Special Function Keys

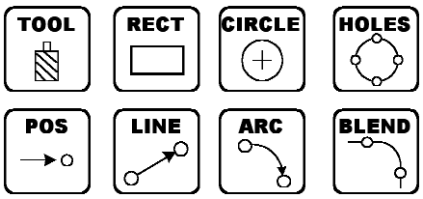
INFO — Access information about any **MILL*PWR*** function.

MM — Switch from inches to millimeters or vice versa.

SETUP — Add to your tool library, set your feed rates, change your display options and define other system parameters.

CALC — Perform standard (+, -, x, ÷), trigonometry, geometry and RPM calculations.

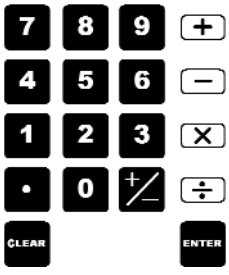
Milling Functions



Use these keys to create a program. All but **BLEND** may also be used as one-time milling functions from the DRO display.

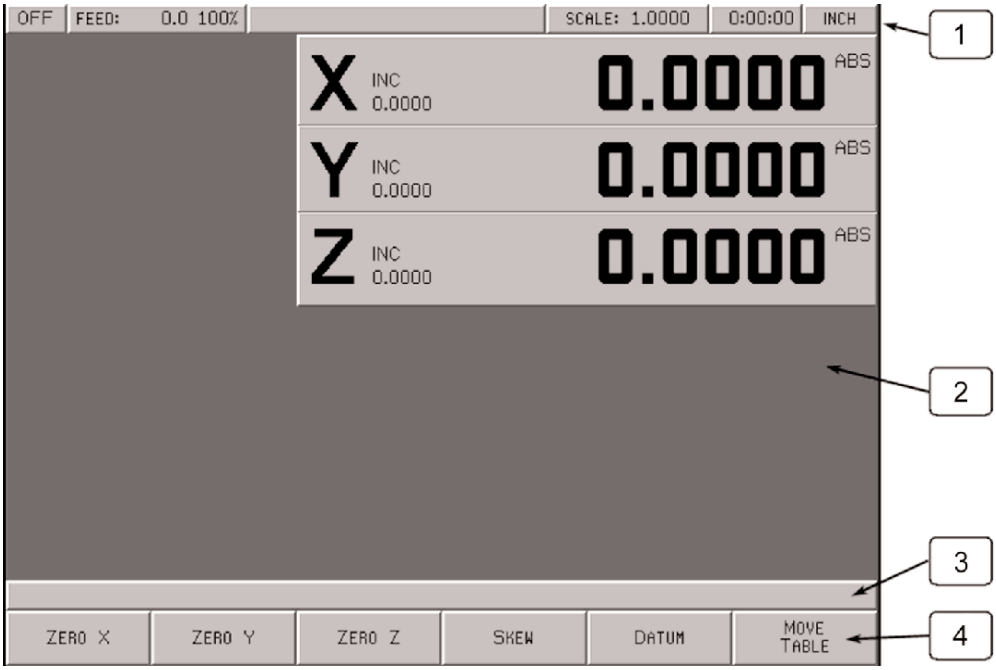
Numeric Keypad and Calculator

Enter program data and perform math calculations. Press the **CLEAR** key to delete information from a data field. Press the **ENTER** key to accept the information you've entered.



Screen Layout

The **MILL*PWR*** display screen is divided into four sections.



1 → **Status bar** - displays the servo motor status (ON/OFF), feed rate, current tool, scale, job clock, and the current display setting (inches or millimeters).

2 → **Information area** - displays information about the job being performed.

- **Readout (DRO)** - used as a digital readout, the display will show the current position for each axis.
- **Program (PGM)** - when programming, a list of program steps (milling functions) and part-view graphics will be displayed.
- **Calculator (CALC)** - the geometry calculator enables you to calculate missing information and then insert it into your program.

3 → **Message line** - operator prompts and messages will appear here.

4 → **Softkeys** - various milling functions appear here; functions are selected by pressing the softkey directly below each category. When a key appears pressed in it is selected. When it appears "up" it is not selected.

Table Stop Button

The large red button located in the lower left corner on the front of your **MILL*PWR*** operator console is the **TABLE STOP**. In the event of a malfunction or programming error, press the **TABLE STOP** button to turn off the servo motors. *This will immediately stop all positioning for each axis.*

WARNING!

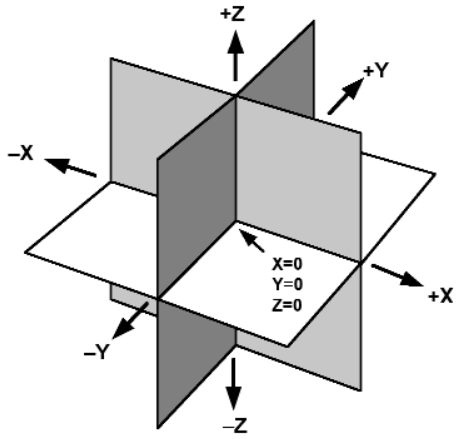
Pressing the **TABLE STOP** button will **NOT** stop the rotation of the cutting tool unless your machine has been configured to do so. If your machine has not been wired to stop the rotation of the cutting tool, be prepared to raise the tool and power down the spindle in addition to pressing the **TABLE STOP** button.

Conventions

Axis Conventions

Count Direction

When programming a part using **MILL*PWR***, table movement and tool movement are determined by the use of positive or negative numbers. **MILL*PWR*** has been factory set with the following positive and negative count directions for the X, Y and Z-axes:



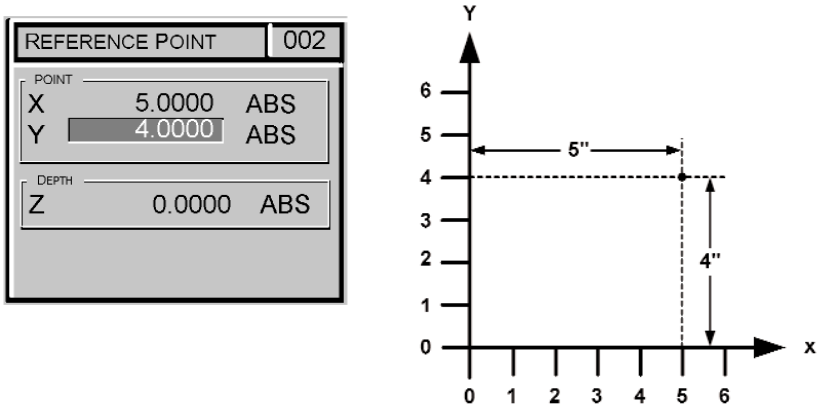
X-axis: The table will move to the left and the tool will move to the right for a positive count direction.

Y-axis: The table will move toward you while the tool moves away from you for a positive count direction.

Z-axis: The quill will move up (away from the table surface) for a positive count direction.

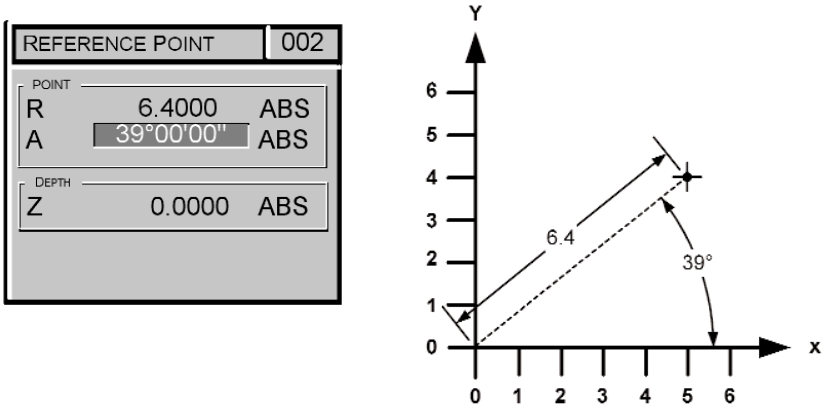
Cartesian Coordinates

A cartesian coordinate is a position that can be measured from the X- and Y-axes.



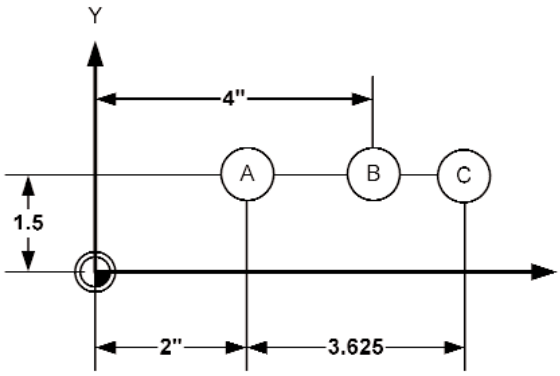
Polar Coordinates

A polar coordinate is a position that is defined by an angle and a radius.



Absolute and Incremental Dimensions

Dimensions that you enter from a print are either absolute or incremental.



Absolute dimensions are measured from datum (also known as workpiece zero). *Incremental dimensions* are measured from one point to another.

Holes A and B are dimensioned using absolute values. Hole C is dimensioned incrementally from Hole A.

When entering these dimensions, we would say:

Hole A: 002 Position/Drill
X 2.0000 ABS
Y 1.5000 ABS

Hole B: 003 Position/Drill
X 4.0000 ABS
Y 1.5000 ABS

Hole C: 004 Position/Drill
X 3.6250 INC 002 - This indicates that the X position will increment from the X value in Step 2.
Y 1.5000 ABS

Note: Both absolute and incremental dimensions may be used to define a position, as shown with Hole C.

It's often easier to describe a location in terms of incremental dimensions rather than calculate its absolute coordinates.

Z-axis Conventions

Z-axis Retract

The Z-axis retract is the position the quill returns to between program steps. By setting a retract position, you can ensure that the tool you are using does not make contact with your workpiece when the quill moves from one position to the next. It's a good idea to establish a retract position for the Z-axis each time you power up your system; otherwise, **MILL*PWR*** will use the quill's upper travel limit as the Z-axis retract position.

Note: Loading a PGM will reset the Z-axis retract to the Z upper limit.

Note: If your tool is *above* the Z-axis retract position when you run a program, the table will rapidly move to position, and then the quill will rapidly move down to the retract position. If the tool is *below* the retract position, the quill will rapidly move up to the retract position first, and then the table will rapidly move into position.

Begin and End Depths

The "Begin" and "End" depths determine where the tool will begin and end its cutting motion. They may be specified as absolute (ABS) or incremental (INC) distances. If the "Begin" field is left blank, you will be prompted to manually position the quill to the desired depth.

Once the quill has reached its retract position, it will rapidly move to the "Begin" depth then move at the programmed feed rate to the "End" depth.

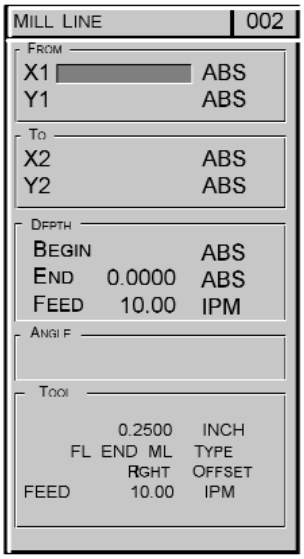
If the travel limit for the Z-axis is set below the established retract position, a travel limit fault will occur and the program will stop.

Pass

"Pass" refers to the number of cuts that are used to machine an area to its "End" depth. You can control how frequently a pass occurs by entering a value in the "Pass" field whenever it appears. (If you don't want to program more than one pass, leave the field blank.)

Disengaging the Z-axis Drive System (Rotary Encoder)

MILL*PWR* provides you with the flexibility to switch between two-axes and three-axes operation.

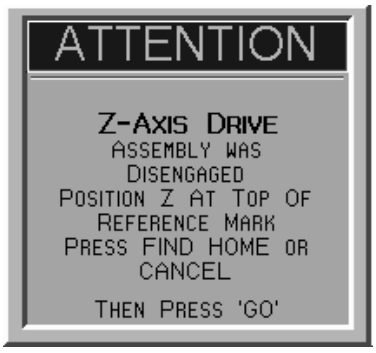


To disengage the Z-axis drive system from your **MILL*PWR*** system:

- Leave the "Begin" field blank when you program a step or a one-time milling function.
- Raise the quill, then loosen the quick release knob located on the front of the Z-axis drive system.
- When you run the program step or one-time milling function, you will be prompted to manually position the quill.

To re-engage the Z-axis drive system, lift the quill handle until you feel the drive assembly become seated, then tighten the quick release knob.

The following Operator Intervention Messages (OIMs) apply to Rotary Encoders:



Disengaging the Z-axis Drive System (Linear Encoder Option)

MILL*PWR* provides you with the flexibility to switch between two-axes and three-axes operation.

To disengage the Z-axis drive system from your **MILL*PWR*** system:

- Leave the "Begin" field blank when you program a step or a one-time milling function.
- Raise the quill, then loosen the quick release knob located on the front of the Z-axis drive system.
- When you run the program step or one-time milling function, you will be prompted to manually position the quill.

To re-engage the Z-axis drive system, lift the quill handle until you feel the drive assembly become seated, then tighten the quick release knob.

The following OIM apply to the optional Z-axis Linear Encoder (only):

Setting Z-axis Datum when Changing Tools

Whenever you encounter a **SET TOOL** step, **MILL*PWR*** will display the DRO screen and let you know which tool to load. If no tool diameter was programmed in the "Tool Step," you will be prompted to provide one.

- Use the **MOVE TABLE** softkey and arrow keys to move the tool away from your workpiece.

- Press the move table softkey again to turn off the motors.
- Insert the required tool into the spindle.
- Using the **MOVE TABLE** feature, position the tool over the surface of a known depth on your workpiece.

- Press the **DATUM** softkey.

- Position your tool so that it touches a known surface, then enter that position into the Z-axis datum.

If the surface is at datum 0, use the Z = 0 softkey.

- Press the **USE** key.

- Raise the tool and press the **GO** key to continue the program.

Drilling Conventions

Peck

"Hole" and "Position/Drill" steps give you the option of programming the quill to "peck" or pause briefly. Peck cycles are used to break chips and reduce chip buildup during drilling operations.

You can control how frequently a peck occurs by entering a value in the "Peck" field.

Tool Retract

Using the "Tool Retract" feature, you can program **MILL*PWR*** to raise the quill at a specified distance or frequency. This will allow the tool you're drilling with to cool.

Dwell

"Dwell" is the length of time (in seconds) that the quill will pause during a tool retract.

Saving, Backing Up, and Creating Directories for Programs

When you create programs with **MILL*PWR***, you can save them in any of three places—within **MILL*PWR***'s internal memory, on a 3½" floppy disk, or on your PC using Remote Storage. Saving your work means it will not be lost if **MILL*PWR*** is powered down or if there is a power interruption.

If you're creating a long program, don't wait until the end to save your work.

Frequent saving reduces the risk of losing your work due to a power interruption.

MILL*PWR* is also equipped with a back up feature that enables you to make duplicate copies of your saved programs to floppy disc or Remote Storage. We recommend that you back up your programs regularly to avoid accidental loss or other problems that may prevent you from recovering your original programs. Backing up your programs takes only a few moments—and will save you valuable time if a problem does occur.

As you save and back up your programs, you can neatly organize them in any of the following three main directories ("MILL*PWR*," "A:" and "REMTSTOR") or in personalized subdirectories that you can create.

> *Note:* Before you save or back up programs onto your PC, refer to **Remote Storage** and **Setup** for instructions.

For more details about how to save programs, back up files and create directories, refer to **Programming**.

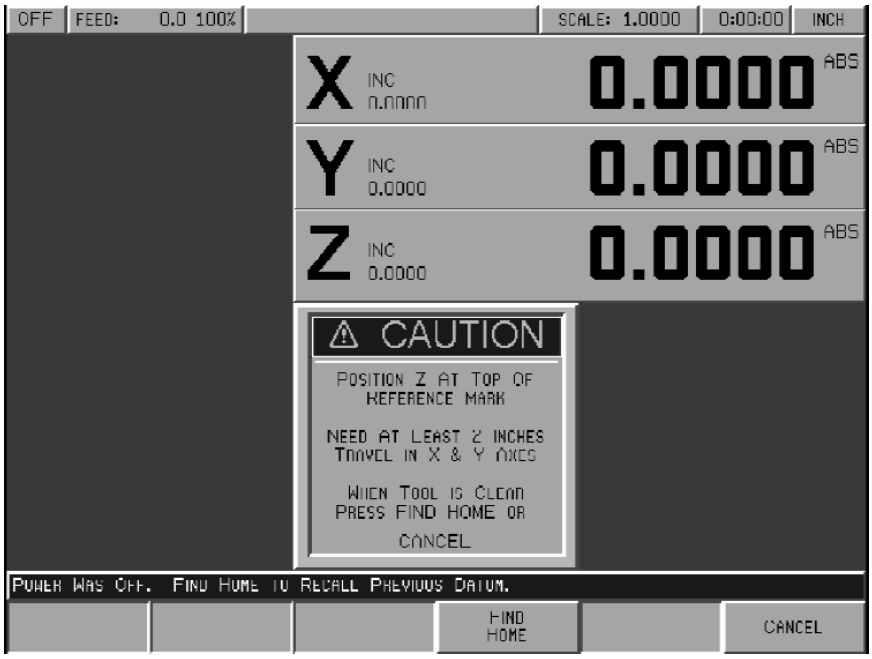
DRO

Start Up

Power Up

Press the power switch (located on the rear of the operator console) to "I." The Power Indicator (located in the upper left corner of the operator console) will light up green.

Once **MILL*PWR*** has been powered up, the following screen will appear:



Screen Saver

Anytime your system is inactive for approximately 90 minutes, the LCD display will shut off, and a blank screen will appear. This screen saver function is designed to prolong the life of your operator console display. If your screen is blank, check that the Power Indicator light is illuminated. Press any key on the operator console or move the table and the display will reappear. If the Power Indicator light is not illuminated yellow, then power to your **MILL*PWR*** system has been interrupted.

Finding Home (Rotary Encoder)

> ⚠ If you don't find home before moving the table, you will risk exceeding the table's travel limits and damaging the milling machine, **MILL*PWR*** or both.

You must find home before you run a program. To find home immediately after startup, locate the Z-axis at the top of the fixed mark then press the **FIND HOME** softkey. Otherwise, press the **DATUM** softkey, then the **FIND HOME** softkey. The table will automatically move a short distance along the Z-, Y- and then X-axes to find home. Z-axis finds home first. You will have to locate the Z-axis at the top of the location mark before finding home.

When finding home, **MILL*PWR*** will use ACU-RITE's advanced Position-Trac™ technology. Position-Trac works by using a very precise distance-encrypted reference mark line pattern that's been placed onto each ACU-RITE precision glass scale included with your **MILL*PWR*** system. Proprietary software decodes the line pattern which then allows you to accurately find home and reestablish workpiece zero from any position.

With Position-Trac, there is no need to leave the system powered up when it is not being used. You'll be able to easily, quickly and accurately reestablish workpiece zero after power loss.

After home has been found, the tool's position (relative to your most recent datum) will be displayed.

Finding Home (Linear Encoder Option)

> ⚠ If you don't find home before moving the table, you will risk exceeding the table's travel limits and damaging the milling machine, **MILL*PWR*** or both.

You must find home before you run a program. To find home immediately after startup, press the **FIND HOME** softkey. Otherwise, press the **DATUM** softkey, then the **FIND HOME** softkey. The table will automatically move a few inches along the Z-, Y- and then X-axes to find home.

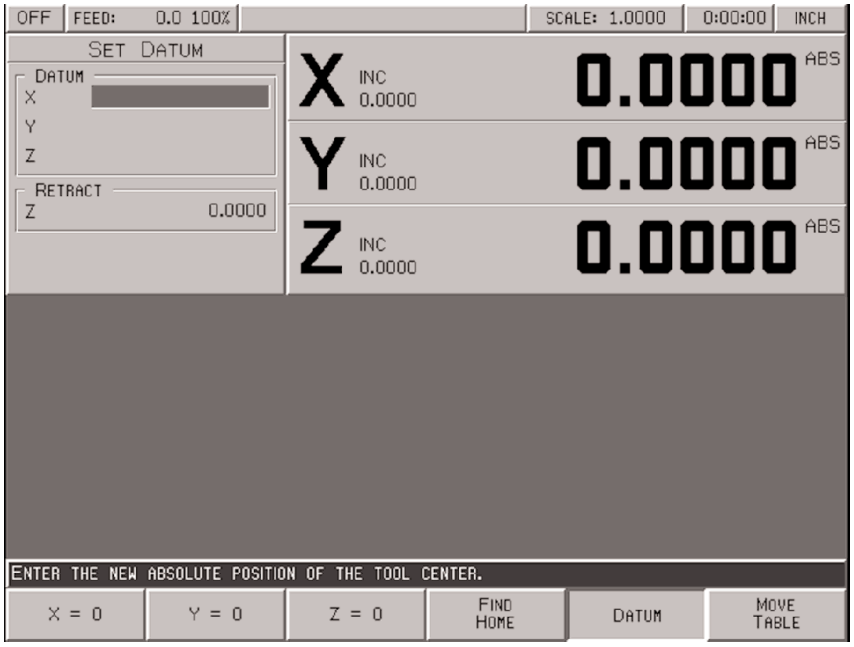
When finding home, **MILL*PWR*** will use ACU-RITE's advanced Position-Trac™ technology. Position-Trac works by using a very precise distance-encrypted reference mark line pattern that's been placed onto each ACU-RITE precision glass scale included with your **MILL*PWR*** system. Proprietary software decodes the line pattern which then allows you to accurately find home and reestablish workpiece zero from any position.

With Position-Trac, there is no need to leave the system powered up when it is not being used. You'll be able to easily, quickly and accurately reestablish workpiece zero after power loss.

After home has been found, the tool's position (relative to your most recent datum) will be displayed.

Datum is a term used to describe "workpiece zero" or "absolute zero."

After you power up and find home, you can quickly move to datum by pressing the POS key, followed by the GO key!



DRO Functions

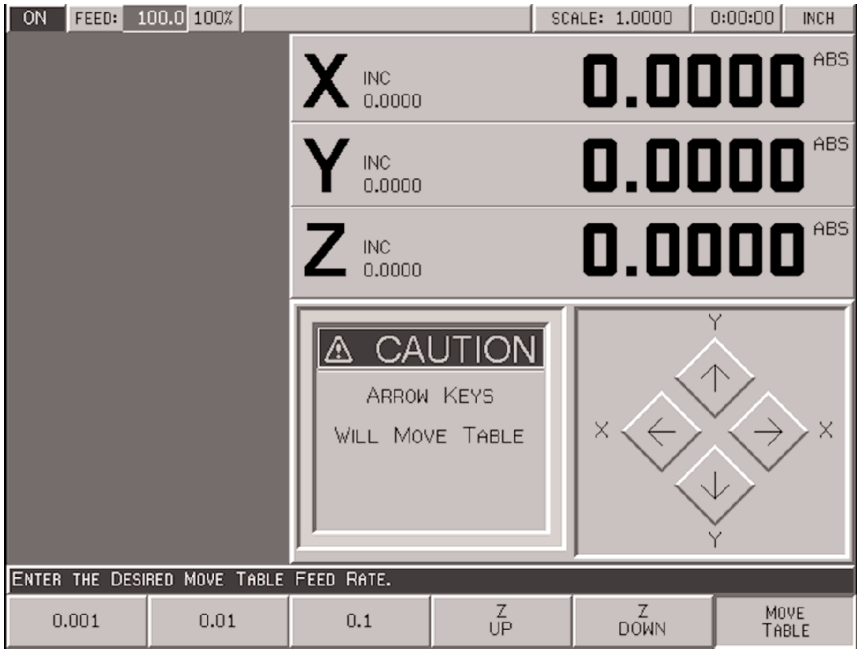
The digital readout (DRO) display shows you the current tool position. While operating in the DRO mode, you can use several functions, such as skew and datum, to set up your job. You can also use this as a standard DRO when you use your machine manually.

Move Table

The move table feature lets you move the table rapidly (or at an established feed rate) using the arrow keys.

- Press the **MOVE TABLE** softkey to turn the servo motors on. Press it again to turn them off.
- Enter the desired feed rate or skip this step to move at a rapid feed rate.
- If you want the arrow keys to move the table in increments, press the **0.001**, **0.01** or **0.1** softkey. (A different set of softkeys will appear if you are measuring in millimeters.)
- Move the table. You can move the X-, Y- and Z-axes simultaneously by pressing two arrow keys and **Z UP** or **Z DOWN** softkeys at the same time.
- Press the **FEED+** and **FEED-** keys to adjust your feed rate.

It's a good idea to fold in the handles before moving the table.



Zeroing an Axis

Pressing the **ZERO X**, **ZERO Y** or **ZERO Z** softkeys will zero the incremental position for those axes.

You need to set datum to establish the point from which all absolute dimensions are based.

Inch/millimeter

You can display inch or millimeter positions. Press the **MM** key to switch from one to the other.

Teach Position

Whenever X, Y or Z coordinates are being entered, the **TEACH POSITION** softkey will appear, enabling you to "teach" **MILL*PWR*** the coordinate(s) you want to use. **MILL*PWR*** will base each coordinate on the current absolute position and enter that position into the field that you've highlighted.

To "teach" **MILL*PWR*** a coordinate (while programming a milling function, such as a line):

- Using the arrow keys, highlight the X-, Y- or Z-axis field.
- Move your tool, indicator, or electronic edge finder to the position you want to teach.
- Press the **TEACH POSITION** softkey to enter that location then press **ENTER**. (If you use an electronic edge finder, the positions will automatically be entered on contact—even if you over-travel.)
- Use the numeric keypad and calculator functions to adjust the number.
- Repeat the steps above for each axis and each location you want to teach.
- Press **USE** to accept the information or press **CANCEL** to return to the previous screen without saving teach position(s).

As you're programming, you can switch between the DRO and PGM screens without losing your program.

Using an Electronic Edge Finder

An ACU-RITE® Electronic Edge Finder enables you to "teach" positions, find the center point of a circle, skew a part or locate datum (also known as workpiece zero) by simply "touching off" on the part. The greatest advantage of an electronic edge finder is that it instantly senses when you've made contact with the point —even when you over-travel.

MILL*PWR* lets you define the "Diameter" and "Unit" of measure (either inches or millimeters) for an electronic edge finder. Once this information has been entered, **MILL*PWR*** will automatically compensate for the radius of the tip of the electronic edge finder when performing any of the operations mentioned above.

To define the diameter and unit of measure:

- Press the **SETUP** key and highlight "Electronic Edge Finder."
- Press the **ENTER** key.
- Enter a value for the edge finder's diameter, select a unit of measure (inches or millimeters) then press the **USE** key.
- Press the **USE NEW SETTINGS** softkey.

Defining the diameter and unit of measure is necessary before using the edge finder because these steps give **MILL*PWR*** the required data it needs to properly place the centerline of the spindle over the indicated edge.

Now the electronic edge finder can be used.

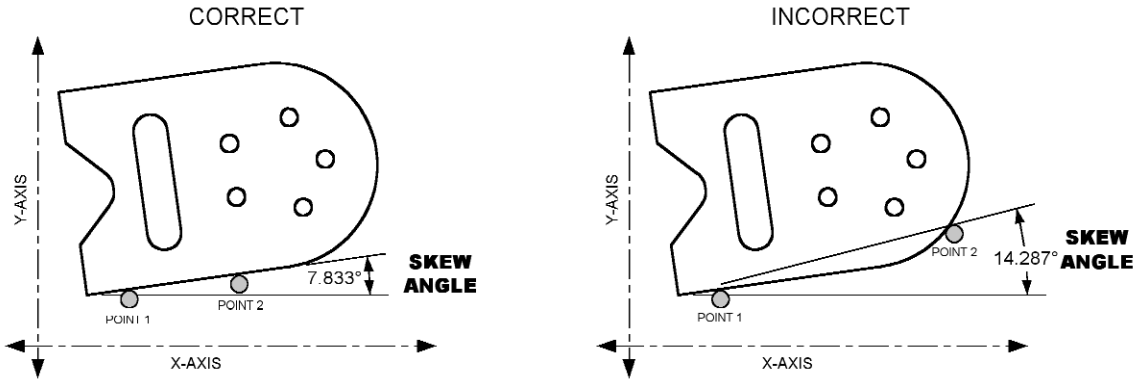
To teach a coordinate with the edge finder, highlight the appropriate field then slowly move the table until the electronic edge finder touches the workpiece. When the electronic edge finder touches the part's surface, the coordinate will appear in the data field. The electronic edge finder is active whenever an entry field is highlighted.

Skewing a Part

With **MILL*PWR***, you can save time setting up a job by skewing your part. The skew function automatically compensates for the offset angle of your part—so if your part is not perfectly parallel with either the X- or Y-axis, you won't have to spend time indicating it in.

To skew a part, simply "touch off" on two or more points along one axis (either X or Y). You can use the electronic edge finder to skew your part or you can use a mechanical indicator and teach position—either way, it's fast and easy.

Note: Choose a line that you want to make parallel with the table's X- or Y-axis—do not enter coordinates along a curve, along two different lines or along a line that's positioned at a true 45 degree angle. **MILL*PWR*** will calculate the skew angle based upon a straight line between the points you've entered.



If you're working with a part that has a rough edge, it's best to enter multiple points along the straightest edge so that **MILL*PWR*** can more accurately calculate the skew angle.

Note: The skew feature does not work with G-code programs. Remove any skew angle prior to running a G-code program.

To skew a part or vise:

Using an electronic edge finder

- Press the **SKEW** softkey.
- Touch off on two or more points along any single straight edge of your part. You'll notice the "Points" and "Angle" change as you enter points. **MILL*PWR*** will calculate the angle to the closest axis line and compensate for the offset of the workpiece.
- Press the **USE** key to accept all of the points and return to the DRO screen. Press the **CANCEL** key to return to the DRO screen *without* accepting any points or affecting your previous skew angle.

Here we used a workpiece as an example. You can also "touch off" on a vise or fixture.

The **CLEAR ANGLE** softkey will reset the number of points and the skew angle to zero.

Using teach position:

- Press the **SKEW** softkey.
- Move the table so that a mechanical indicator rests against any straight edge on the part. Press the **TEACH POSITION** softkey to enter your coordinate. You'll notice that the "Points" change.
- Now move the table so that the mechanical indicator touches another point on the same straight edge. Press the **TEACH POSITION** softkey. You'll notice that the "Points" and "Angle" change.

  Repeat this process for any additional points. **MILL*PWR*** will calculate the angle to the closest axis line and compensate for the offset of the workpiece.
- Press the **USE** key to accept all of the points and return to the DRO screen. Press the **CANCEL** key to return to the previous screen *without* affecting your previous skew angle.

  The **CLEAR ANGLE** softkey will reset the number of points and the angle to zero.

Establishing Datum

Datum, also known as workpiece zero or absolute zero, is a point of reference that **MILL*PWR*** bases all of your part's coordinates from.

Datum will need to be established for every job. Datum's location may be indicated on your print; if it's not, establish a datum that allows you to enter most of the part's dimensions directly, with the least amount of calculations.

When establishing datum, you may find it easiest to locate a known point on each axis, such as the edge of the part or a location on the vise or fixture.

Refer to the procedure below as a basic guide for establishing datum. You may decide to "touch off" using an electronic edge finder, a mechanical edge finder or a tool. Datum may be set at a point on the top surface or a position above or beneath the surface. X and Y datum may be set on an edge, or offset into or off of an edge, or where there's no material present (such as in the center of a circular part) do what's easiest for your particular job.

MILL*PWR* will retain datum even after your system has been powered down.
See “Recalling A Datum”.

Note: When the “Datum” menu appears, you will also be asked to establish a Z-axis retract position. The Z-axis retract position is the position that the quill returns to between program steps (refer to **Conventions** located in Section 1).

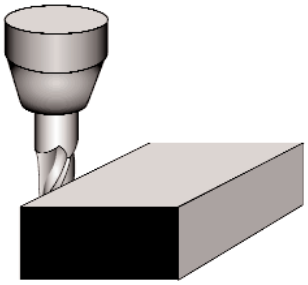
To establish datum:

Where and how datum is established will vary from job to job. The following is one of the most common methods of establishing datum. These basic principles can be applied when setting datum for your parts, making adjustments to the procedure as needed.

First establish datum at the corner where the left, front and top surfaces of the part intersect. This is accomplished by "touching" each face with the tool that is being used for cutting.

Define datum one axis at a time. Begin here with the X-axis:

- From the DRO screen, press the **DATUM** softkey.
- Insert the tool.

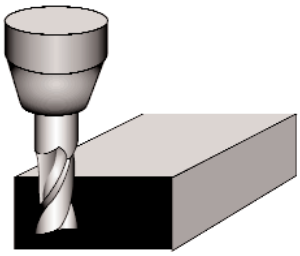


- Position the tool so that it is near, but not touching, the left side of the part.
- Lower the tip of the tool so that it falls below the top surface of the part.
- Slowly move the table along the X-axis, spinning the tool by hand as you go. Pay close attention as the tool approaches the part—you'll feel a subtle bump when they come into contact. **Stop the table at the moment the tool touches the part.**
- Using the keypad, enter the radius of the tool (the distance from the center of the tool to the edge of your part). Be sure to specify if it's a negative value.

  Note: In this example, a negative value will need to be specified because the tool's center is on the negative side of the datum (refer to **Axis Conventions** located in Section 1).

Use the MOVE TABLE softkey to help with long table moves.

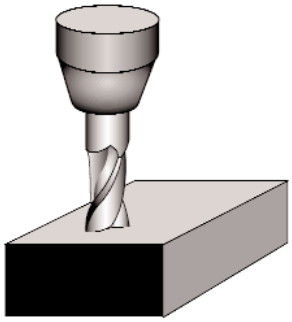
- Press the **ENTER** key.



Now set datum for the Y-axis using the same procedure:

- Position the tool so that it is near, but not touching, the front face of the part. The tip of the tool should fall below the top surface of the part.
- Slowly move the table along the Y-axis, spinning the tool by hand as you go. Pay close attention as the tool approaches the part—you'll feel a subtle bump when they make contact with each other. **Stop the table at the moment the tool touches the part.**
- Using the keypad, enter the radius of the tool into the"Y" field (be sure to specify if it is a negative value).
- Press the **ENTER** key.

Setting datum for the Z-axis:



- Position the tool so that its tip touches the top surface of the part.
- Using the keypad, enter "0" into the "Z" field (or press the Z = 0 softkey).
- Press the **ENTER** key.
- Highlight the “Z Retract” field.
- Either:

  Enter the Z-axis retract position (the position that you want the quill to return to between steps—it must be located above the top surface of the part); OR

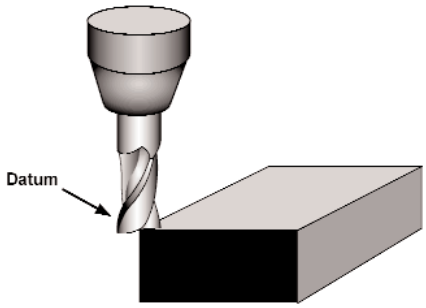
  Raise the quill to the desired retract position, then press the **TEACH** softkey.

- Press the **USE** key.

Datum and the Z-axis retract position have now been established.

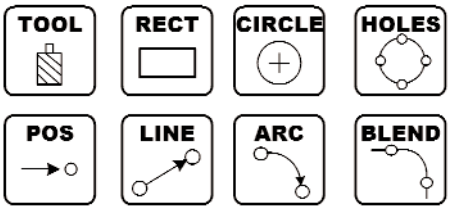
You can quickly move to Datum for X and Y by pressing the POS key. Check that your position is 0.0000 ABS for both X and Y, then press the GO key.

It's a good idea to test the datum setting before beginning programming. To confirm that the new datum is correct:



- Raise the tool and move the table until both the X- and Y-axes displays read "0.0000."
- Lower the tool until it touches the part.
- Check the tool's position. The lower left corner of the part should be located directly beneath the center point of the tool.
- Now check the readout. If the Z-axis display says "0.0000," then the datum is accurate and programming can begin. If a value other than 0.0000 appears in the Z-axis display, repeat the procedure for establishing datum.

Hard Key Milling Functions



Most of the hard key milling functions can be used individually as one time milling routines. That means you can use these keys without creating a program.

The only hard key milling function you can't use as a one-time milling function is **BLEND**. The blend function inserts a connecting radius between two features (steps) in a program.

Hard key milling functions are ideal for jobs that only require one operation. You provide the required information once, and **MILL*PWR*** will "remember" it for each piece you machine.

Example:

To drill the same bolthole pattern on several identical parts, instead of creating a one-step program, use the **HOLES** hard key milling function from the DRO screen.

- First, setup the tool and workpiece. Establish the skew angle (if any), datum and the tool retract position.
- From the DRO display, press the **HOLES** key, and select which hole pattern you want to drill from the available softkeys—**ROW, FRAME, ARRAY,** or **BOLT CIRCLE**.
- Enter the required information and press the **GO** key to machine the first part.
- Change the part. Press the **HOLES** key, then the pattern.

MILL*PWR* will automatically refer to the data you've entered for each part thereafter.

To change the hole pattern size, depth, location, number of holes, etc., press the **HOLES** key again. Now press the appropriate softkey, enter the new information and then press the **GO** key.

This also applies to rectangles, circles, lines and arcs. The rectangle and circle milling functions require you to establish a tool offset. Lines and arcs only require a tool offset if the tool follows the left or right edge. It's a good idea to setup the tool before using either of these function keys. (Refer to **Program Steps** for a complete description of each function.)

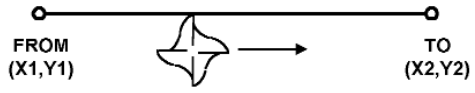
Note: The "Tool" setting on **MILL*PWR***'s "Status" bar (located along the top of the **MILL*PWR*** screen) will indicate which tool has been selected. If there's no tool identified, or if it's incorrect, you'll need to start with a "Set Tool" step that accurately identifies the tool you're using (refer to **Programming A Tool Step**).

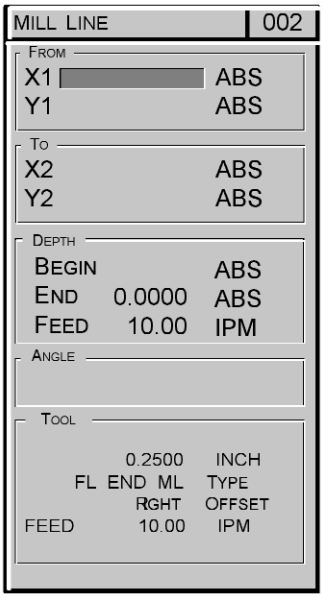
PROGRAMMING

Programming Considerations

"From" and "To" Points

Lines and arcs are defined by their “From” point (the point where they begin) and “To” point (the point where they end).

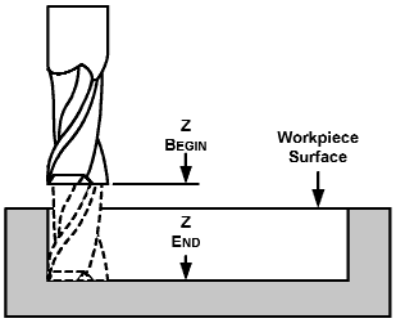




Depth of Cut

When you’re programming the depth of cut, you’ll be prompted to provide the “Begin” and “End” locations for the Z-axis.

The location that you enter into the “Begin” field tells **MILL*PWR*** where you want the quill to begin cutting at the programmed feed rate. The “End” location defines the depth of the cut.

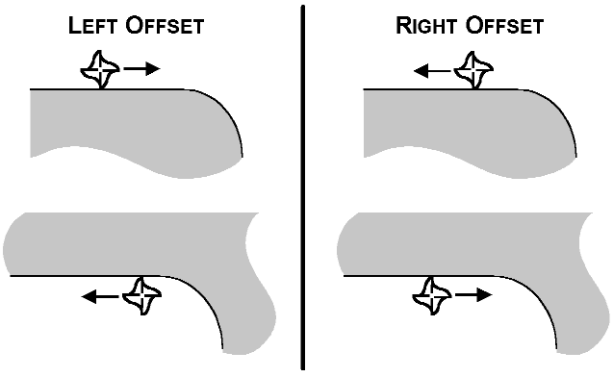


Always check that the “Begin” location is above the surface of the workpiece.

Pass

“Pass” refers to the number of cuts that are used to machine an area to its “End” depth. You can control the number of passes by entering a value in the “Pass” field whenever it appears. (If you don’t want to program more than one pass, leave the field blank.)

Tool Offset



With **MILL*PWR***, you never have to calculate the actual tool path. By using left and right offsets, you can program the dimensions of the part as identified on your print.

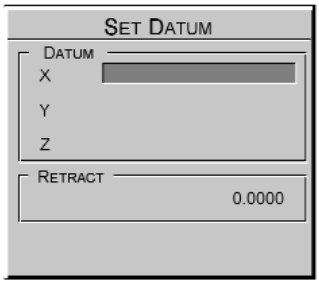
When you program a line, arc, frame, etc., use the “Tool Offset” field to tell **MILL*PWR*** which side of the line you want the tool to be on.

To determine which offset to use, picture yourself following the tool as it is moving. If the tool needs to be on the left side of the line, use a "left" offset. If the tool needs to be on the right side of the line, use a "right" offset.

If you use a "center" offset, the programmed dimensions are for the center of the tool.

For some milling functions, like frame and arc, "inside" and "outside" offsets are available to make it easier for you to define your tool offset.

Datum Selection



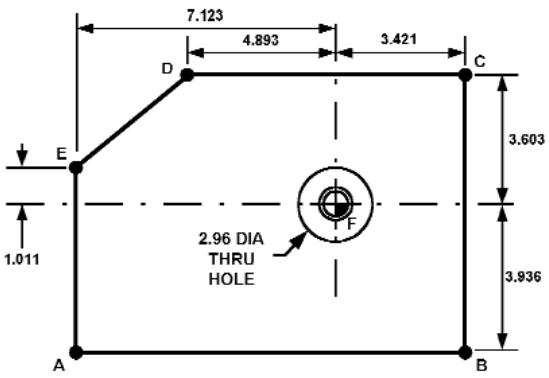
Datum is where workpiece (absolute) zero is located. If datum isn't defined on your print, then determine datum based upon where most of your dimensions originate. You should pick a point which will let you enter most of the dimensions directly, with few (if any) calculations (refer to **Establishing Datum** located in Section 2).

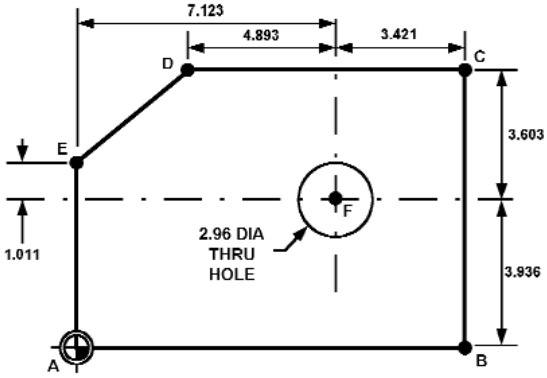
As you establish datum, you’ll be prompted to provide a Z-retract position (the position the quill returns to between program steps). By setting a retract position, you can ensure that the tool you are using does not make contact with your workpiece when the quill moves from one position to the next. It’s a good idea to establish a retract position for the Z-axis each time you power up your system; otherwise, **MILL*PWR*** will use the quill’s upper travel limit as its default retract position (refer to **Conventions** located in Section 1).

Absolute vs. Incremental Dimensions

MILL*PWR* allows you to enter both absolute and incremental dimensions. A dimension measured from the point you defined as datum is an absolute dimension. A dimension measured from any other point is an incremental dimension.

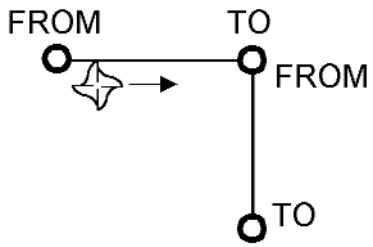
In the examples below, the print on the left shows datum located at the center of Hole F—all dimensions are absolute. The print on the right shows datum located in the lower left corner—point A. Most of these dimensions are incremental.





Continuous Milling

When you program a continuous contour of lines and/or arcs, **MILL*PWR*** will cut the contour without stopping. **MILL*PWR*** will automatically recognize continuous contours as you're programming. There are no special key presses or other functions to learn.



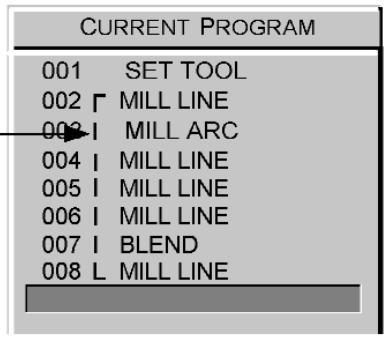
For lines and arcs to be continuous, they must:

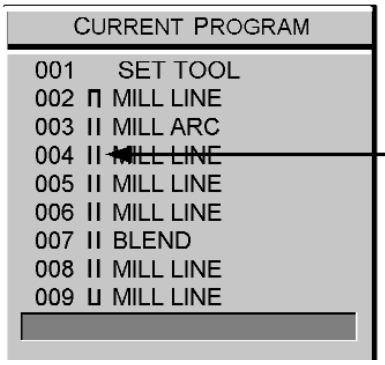
- Be consecutive steps in a program
- Have the same depth
- Be cut with the same tool
- Be cut using the same tool offset
- Share a common “From” or “To” point (one step must end at the point where another begins)

If one step follows another, **MILL*PWR*** assumes that you want them to be connected. It automatically fills in the "From" point, "Depth," and "Tool Offset." All you have to do is fill in the "To" point and press **USE**.

Note: **MILL*PWR*** will allow you to program different feed rates within each step of a continuous contour.

Single lines indicate an open-ended continuous tool path.





Double lines indicate a closed continuous tool path.

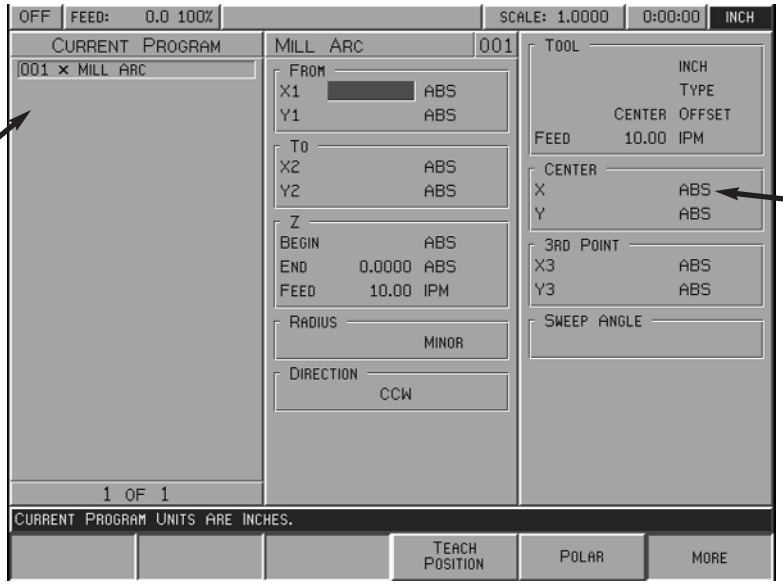
Note: An "X" before or after a step number indicates that the step is invalid. Highlight the step, press enter and correct the information as needed. Press use when finished.

Creating a Program

- Press the **PGM** key, and the following program screen will appear.

Programs are created by developing a list of milling steps to be performed. As you add to your list, each step will immediately be drawn on the screen so that you can see a graphic display of your part in progress.

- To enter a milling step, press the appropriate hard key milling function (such as Tool). The milling function keys are the eight yellow keys located in the upper right corner of your keypad. The function you select will appear in the program listing and will enable you to enter the information describing the step into the program.

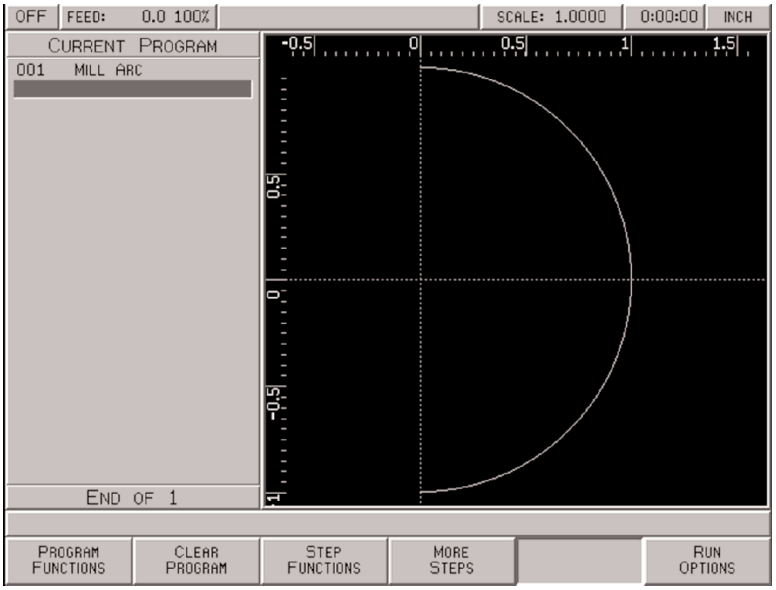


Program steps are added here.

Enter information about a milling function—such as an arc—into forms like this.

- After entering all the data for a step, press the **USE** key to add the step to your program. This immediately updates the part graphic and positions the cursor for the next step.

If you decide not to finish a milling function that you have begun, simply press the **CANCEL** key.



- To edit a step, use the arrow keys to highlight the step you want to change and press **USE** or **ENTER**. When you have made your changes, press **USE** to accept your changes and place the step back into your program.

- To delete a step, highlight the step you want to delete, then press **CLEAR**.

- To insert a step between the two existing steps, position the cursor to where you want the new step to go, and press the desired milling function key.

In addition to the hard key milling functions, the **MORE STEPS** softkey lets you pick from a number of other useful steps, such as **CUSTOM POCKET**, **ISLAND, ELLIPSE**, **MIRROR**, **REPEAT** and **ROTATE**—each of which are described in the **Program Steps** section of this manual.

The View Key

If you need to see your part-graphic in more detail, press the **VIEW** key. This enables you to access the following softkeys:

The **FOLLOW TOOL**, **SHOW TOOL PATH** and **ZOOM** functions may be used simultaneously. Press the **VIEW** key (or the **CANCEL** key) at any time to return to the PGM screen.

FOLLOW TOOL

Press both the **FOLLOW TOOL** and **ZOOM IN** softkeys to see a close-up of the tool's path. **MILL*PWR*** will automatically adjust the part graphic so that the tool is always in view.

SHOW TOOL PATH

The **SHOW TOOL PATH** softkey shows the tool's cutting path as you run the program. With this feature enabled, you can see where the tool has been.

ZOOM IN, ZOOM OUT and RESTORE

The **ZOOM IN** softkey will magnify the part graphic. The arrow keys will enable you to adjust the view up, down, left and right. The **ZOOM OUT** softkey will de-magnify your part graphic. **RESTORE** will return the part graphic to its original size.

Running a Program

There are a few things you'll need to do before running a program, such as skewing the part and establishing datum.

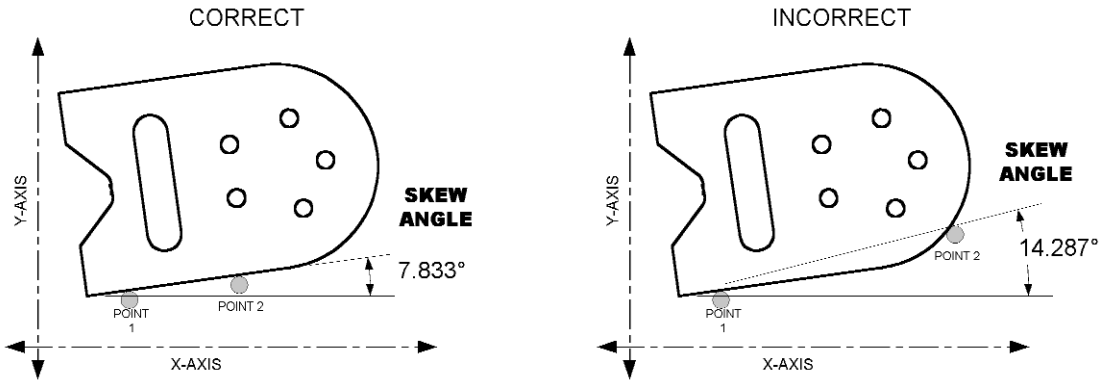
Skewing a Part

Note: It is important to skew a part prior to establishing datum for accuracy.

With **MILL*PWR***, you can save time setting up a job by skewing your part. The skew function automatically compensates for the offset angle of your part—so if your part is not perfectly parallel with either the X- or Y-axis, you won't have to spend time indicating it in.

To skew a part, simply "touch off" on two or more points along one axis (either X or Y). You can use the electronic edge finder to skew your part or you can use a mechanical indicator and teach position—either way, it's fast and easy.

Note: Choose a line that you want to make parallel with the table's X- or Y-axis—do not enter coordinates along a curve, along two different lines or along a line that's positioned at a 45 degree angle. **MILL*PWR*** will calculate the skew angle based upon a straight line between the points you've entered.



If you're working with a part that has a rough edge, it's best to enter multiple points along the straightest edge so that **MILL*PWR*** can more accurately calculate the skew angle.

Using an electronic edge finder:

- From the DRO screen, press the **SKEW** softkey.
- Touch off on two or more points along any single straight edge of your part. You'll notice that the "Points" and "Angle" change as you enter points.
- Press **USE** to accept all of the points and return to the DRO screen. Press **CANCEL** to return to the DRO screen *without* accepting any points or affecting your previous skew angle.

  The **CLEAR ANGLE** softkey will reset the number of points and the skew angle to zero.

Using teach position:

- From the DRO screen, press the **SKEW** softkey.
- Move the table so that a mechanical indicator rests against any straight edge on the part. Press the **TEACH POSITION** softkey to enter your coordinate.You'll notice that the "Points" change.
- Now move the table so that the mechanical indicator touches another point on the same straight edge. Press the **TEACH POSITION** softkey. You'll notice that the "Points" and "Angle" change.

  Repeat this process for any additional points.
- Press **USE** to accept all of the points and return to the DRO screen. Press **CANCEL** to return to the previous screen *without* affecting your previous skew angle.

  The **CLEAR ANGLE** softkey will reset the number of points and the angle to zero.

Establishing Datum

Datum, also known as workpiece zero or absolute zero, is a point of reference that **MILL*PWR*** bases all of your part's coordinates from.

You will need to establish datum for every job. Datum's location may be indicated on your print; if it's not, then establish a datum that allows you to enter most of your part's dimensions directly, with the least amount of calculations.

When establishing datum, you may find it easiest to locate a known point on each axis, such as the edge of your part or a location on your vise or fixture.

Refer to the example below as a basic guide for establishing datum. You may decide to "touch off" using an edge finder instead of a tool. Datum could be set at a point on the top surface, a position beneath the surface, or at a point where there's no material present (such as in the center of a circular part). The possibilities are endless—do what's easiest for your particular job.

MILL*PWR* will retain the datum you've set after your system has been powered down.

Note: When the “Datum” menu appears, you will also be asked to establish a Z-axis retract position. The Z-axis retract position is the position that the quill returns to between program steps (refer to **Conventions** located in Section 1).

To establish datum:

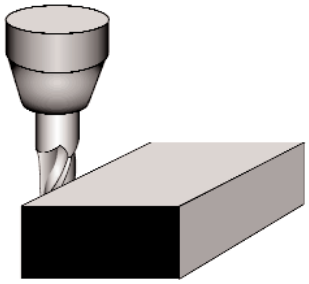
As we’ve already mentioned, where and how you establish datum will vary from job to job. Here we’ll walk you through one of the most common methods of establishing datum. Once you’ve learned the basics, apply the same principle when setting datum for your own parts, making adjustments to the procedure as needed.

We’ll establish datum on the corner where the left, front and top surfaces of our part intersect. We’ll accomplish this by “touching” each face with the tool that we’re planning to cut the part with.

Define datum one axis at a time. Here we’ll begin with the X-axis:

- From the DRO screen, press the **DATUM** softkey.
- Insert the tool you are planning to cut the part with into the spindle.

- Position the tool so that it is near, but not touching, the left side of your part.



- Lower the tip of the tool so that it falls below the top surface of the part.
- Move the table along the X-axis, slowly spinning the tool by hand as you go. Pay close attention as the tool approaches the part—you'll feel a subtle bump when they come into contact. **Stop the table at the moment the tool touches the part.**

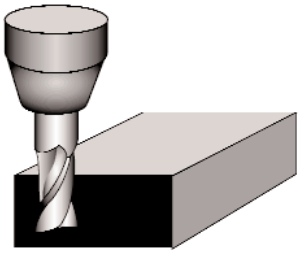
Use the MOVE TABLE softkey to help with long table moves.

- Using the keypad, enter the radius of the tool (the distance from the center of the tool to the edge of your part) into the "X:" field. Be sure to specify if it's a negative value.

  Note: In our example, we'll need to specify a negative value, because the tool's center is on the negative side of our datum (refer to **Axis Conventions**).

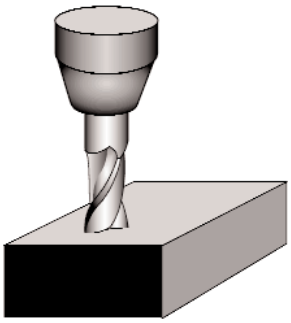
- Press the **ENTER** key.

Now we'll set datum for the Y-axis using the same procedure:



- Position the tool so that it is near, but not touching, the front face of your part. The tip of the tool should fall below the top surface of the part.
- Move the table along the Y-axis, slowly spinning the tool by hand as you go. Pay close attention as the tool approaches the part—you'll feel a subtle bump when they make contact. **Stop the table at the moment the tool touches the part.**
- Using the keypad, enter the tool's radius into the"Y:" field (be sure to specify if it is a negative value).
- Press the **ENTER** key.

Finally, we'll set datum for the Z-axis:



- Position the tool so that its tip touches the top surface of your part.
- Using the keypad, enter "0" into the "Z:" field (or press the Z = 0 softkey).
- Press the **ENTER** key.
- Highlight the “Z Retract” field.
- Either:

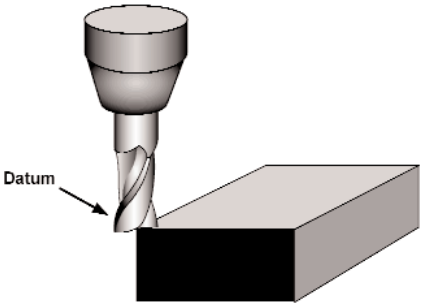
  Enter the Z-axis retract position (the position that you want the quill to return to between steps—it must be located above the top surface of your part); OR

  Raise the quill to the desired position, then press the **TEACH** softkey.
- Press the **USE** key.

Datum and the Z-axis retract position have now been established.

You can quickly move to datum for X and Y by pressing the POS key. Check that your go to position is 0.0000 for both X and Y, then press the GO key.

It's a good idea to test your datum setting before you begin programming. To confirm that your new datum is correct:



- Raise the tool and move the table until both the X- and Y-axes dis plays read "0.0000."
- Lower the tool until it touches your part.
- Check the tool's position—the lower left corner of the part should be positioned directly beneath the center point of the tool.
- Now check the readout screen. If the Z-axis says "0.0000," then your datum is accurate and you can begin programming. If a value other than 0.0000 appears in the Z-axis display, repeat the procedure for establishing datum.

Testing Your MILL*PWR* Program

> ⚠ Whenever you are about to run a program, check that the handles are recessed.

Before you machine a part, it is always a good idea for you to test your program for things like the correct tool path, count direction, feed rate, and sequence of operations. **MILL*PWR*** provides several run-time options to assist you. From the PGM screen, press **RUN OPTIONS** to display the following softkeys:

Press any softkey to activate the option; press it again to deactivate it.

Note: Before you press the **GO** key to begin the "Single Step," "Dry Run" or "Manual Positioning" features, check that the tool will not touch the workpiece when the quill begins to move. To avoid interference, we suggest that you do one or more of the following:

- Lower the knee
- Remove the tool or workpiece
- Reestablish datum away from the part (refer to **Establishing Datum**)

SINGLE STEP

Normally, a continuous contour will be machined without stopping. With "Single Step" activated, **MILL*PWR*** will stop after each step. This enables you to check the position of the Z-axis relative to your part and ensure that the tool path and other program details are correct. Press the **GO** key to begin.

To test your program quickly, press both the DRY RUN and GRAPHICS ONLY softkeys.

DRY RUN

With "Dry Run" activated, **MILL*PWR*** will run your entire program at high speed without stopping. You can visually follow the position of the tool relative to your part and ensure that the tool path and other program details are correct. The dry run speed is defined in **Setup**. Press the **GO** key to begin.

GRAPHICS ONLY

With this activated, the table and quill will not move. The graphics screen will show you how your part will be cut. You can see all the normal feed rates, tool changes and so on. Press the **GO** key to begin.

Note: Dry Run and Graphics Only can be used to quickly verify your program.

MANUAL POSITIONING

Use this option if you want to position the table using the handles. **MILL*PWR*** will operate just like a programmable readout—each target position will be preset into the readout, and you will be prompted to position the table manually. This feature is especially useful when you're navigating around islands. Press the **GO** key to begin.

DISABLE LOOK AHEAD

Normally, **MILL*PWR*** checks each step in a continuous tool path with other steps to determine if there is an intersection in the tool's cutting path (such as in the number "8"). This is called "look ahead." If you press the **DISABLE LOOK AHEAD** softkey, your program will run *without* performing this function. Press the **GO** key to begin.

Machining Your Part

> ⚠ Whenever you are about to run a program, check that the handles are recessed.

Before you run a program step, check the "Status" bar (located along the top of the **MILL*PWR*** screen) to ensure that the tool identified by **MILL*PWR*** matches the tool in the spindle. If there's no tool identified, or if it's incorrect, you'll need to start with a "Set Tool" step that accurately identifies the tool you're using (refer to **Set Tool**).

To move quickly to a step, key in its number (look in the message bar), and press ENTER.

After you've highlighted the step that you want to begin with, press the **GO** key. An Operator Intervention Message (OIM) will ask you to confirm this tool is correct. Check that the correct tool is being used, then press the **GO** key again to begin milling.

If your tool is positioned *above* the Z-axis retract position before you begin to run a program, the table will rapidly move to position, and then the quill will rapidly move to the retract position. If the tool is *below* the retract position, the quill will move first.

The remote STOP/GO switch acts as PAUSE if the table is moving and as GO if the machine is paused or stopped.

Once the quill has reached its retract position, it will rapidly move to the "Begin" depth then move at the programmed feed rate to the "End" depth (refer to **Begin and End Depths**).

If the travel limit for the Z-axis is set below the established retract position, a travel limit fault will occur and the program will stop.

MILL*PWR* will automatically pause at points that require you to take action (e.g., change tools). After each task has been completed, press the **GO** key.

If you press the **STOP** key once while cutting, your tool will pause in its cutting path and an OIM will appear. Press **GO** to resume machining, or **STOP** again to end the program.

Note: If you wish to start machining from the middle of a program and the tool in your spindle does not match the tool information displayed in the current tool field, you must start the program at the programmed tool step for that tool.

Feed+ and Feed-

The **FEED+** and **FEED-** keys will change your feed rate by a certain percentage with each key press. The feed rate percentage will be displayed in the status bar at the top of the screen. A feed rate percentage of 100% means that actual feed rates will run at 100% of the programmed feed rates. If the feed rate percentage is 50%, actual feed rates will run at half of the programmed feed rates.

You can press the **FEED+** and **FEED-** keys at any time, even while the table or quill is moving.

Manually Positioning the Quill (Optional Linear Encoder Only)

Programs that do not include a “Begin” depth will require you to manually position the Z-axis during machining. The same is true when the Z-axis has been disengaged during setup (refer to **Setup**).

If you have programmed an “End” depth, when it is time for you to manually position the Z-axis, **MILLPWR** will preset the value into the readout's Z-axis display. The DRO screen will appear, along with the prompts shown below.

MILLPWR is factory set in a “distance to go” display view. This way, any dimension you have programmed will be "preset" into the readout display. Every move will end at zero.

If you wish, you can setup **MILLPWR** to display incremental travel (refer to **Display Options** in Section 7). In this view, every move will end at the programmed depth.

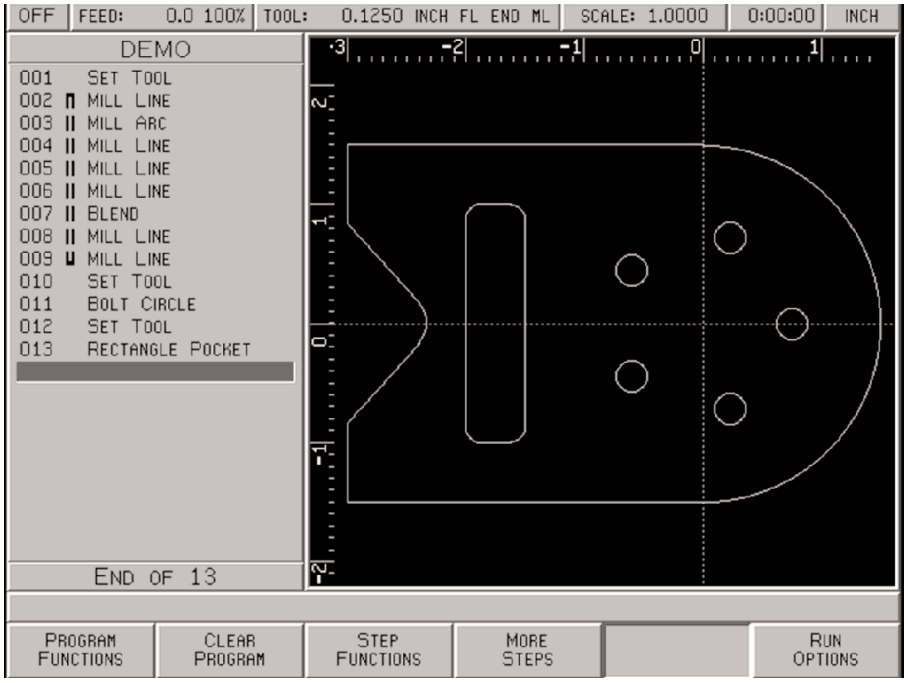
This Operator Intervention Message will appear for optional Z-axis Linear Encoder systems only:

Program Functions

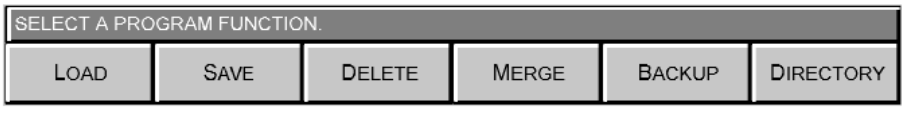
Accessing Load, Save, Delete, Merge, Backup and Directory Options

MILL*PWR* offers several versatile features for loading, saving, deleting, merging and backing up your programs. You can also easily organize your programs by creating directories.

To access these features, from the PGM screen, press the **PROGRAM FUNCTIONS** softkey.



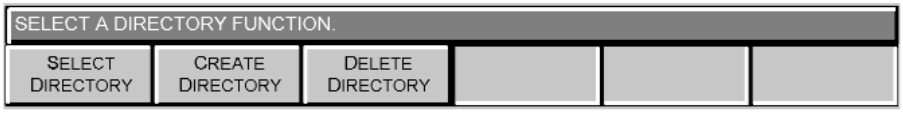
You may load a program, save the program you've been working with, delete programs, merge a saved program into your current one, create a backup copy, or select, create or delete a directory.



Directories

One of the best ways to keep your programs organized is to save them in directories. Directories are like file folders—they should be clearly labeled and contain closely related programs. They may be used to group programs by job, operator, date, customer, or any other method you prefer.

- Press the **PROGRAM FUNCTIONS** softkey, then press the **DIRECTORY** softkey.



Now you can open an existing directory, create a new directory, or delete a directory that you no longer need.

Note: A directory can only be deleted if it is empty. You must first delete or move all part programs from a directory prior to deleting them. See **Deleting a Program** located in Section 3.

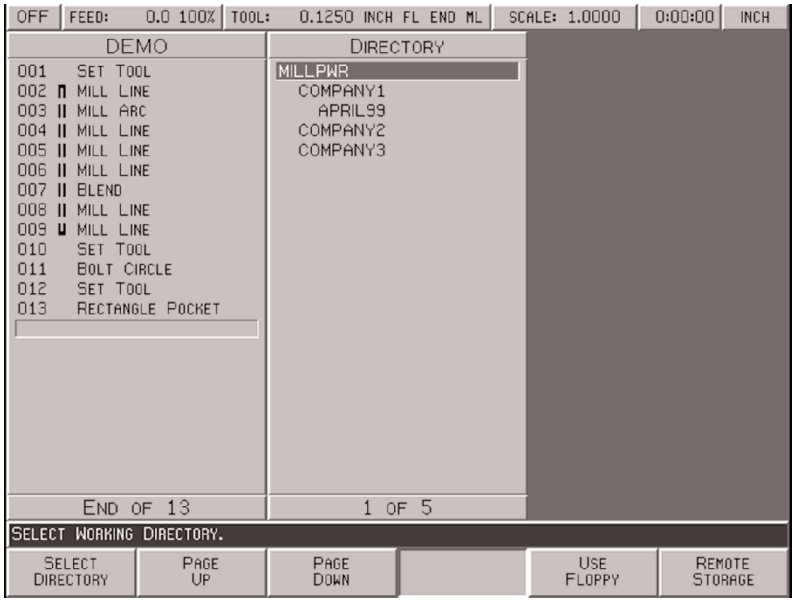
Creating a Subdirectory

We recommend creating matching directories on MILLPWR and your PC. This will make it easier to keep your original programs and backup programs organized.

The best approach to take when creating a subdirectory is to decide first where to place it. You can place it on the "MILLPWR directory," on a floppy disk ("A:") directory or on your PC ("REMTSTOR") directory, or you can place it within subdirectories that you have already created.

In the example below, we've created four subdirectories in the **MILLPWR** folder. Three of the subdirectories are named for our top customers: COMPANY1, COMPANY2 and COMPANY3.

"COMPANY1" has placed several part orders for a single month. To help us find those part programs quickly and easily, we created the subdirectory "APRIL99" to save them in.



If we want, we can create another subdirectory under APRIL99, layer another one under that, one under that and so on. The number of directories you create and how you layer them is up to you.

To create directories for your programs**:**

- Press the **PGM** key, the **PROGRAM FUNCTIONS** softkey and then the **DIRECTORY** softkey. Now press the **SELECT DIRECTORY** softkey, and a directory list will appear.

- Highlight the directory where you want to store your new subdirectory.

  Select **MILL*PWR***'s *internal memory:*

  "MILLPWR" and any subdirectories should appear under the "Directory" heading. (If “MILLPWR” does not appear, check that the **USE FLOPPY** and **REMOTE STORAGE** softkeys are *not* selected.)

  Select a 3½" floppy disk:

  Press the **USE FLOPPY** softkey. "A:" and any subdirectories should appear under the "Directory" heading.

  Select your PC:

  Press the **REMOTE STORAGE** softkey. "REMTSTOR" and any subdirectories should appear under the "Directory" heading.

The REMOTE STORAGE softkey will only appear if* MILL*PWR has been set up for remote storage.

- Using the arrow keys, highlight the existing directory you want to put your new subdirectory in. (In our example, we highlighted "MILLPWR," then created a subdirectory entitled "COMPANY1.")

- Press the **SELECT DIRECTORY** softkey again to verify your choice. The "Directory" screen will disappear.

- Now press the **DIRECTORY** softkey.

- Press the **CREATE DIRECTORY** softkey. You will be asked to name your directory.

- You can name your directory using the numeric keys on the operator console or by selecting letters from the **ALPHABET** option. Use the arrow keys to move from letter to letter then press the **ENTER** key to make a selection. Program names are limited to eight characters, consisting of numbers and/or letters.

- After you have named your directory, press the **CREATE DIRECTORY** softkey again to enter your choice. The "Directory" screen will now disappear.

Additional directories may be added at any time.

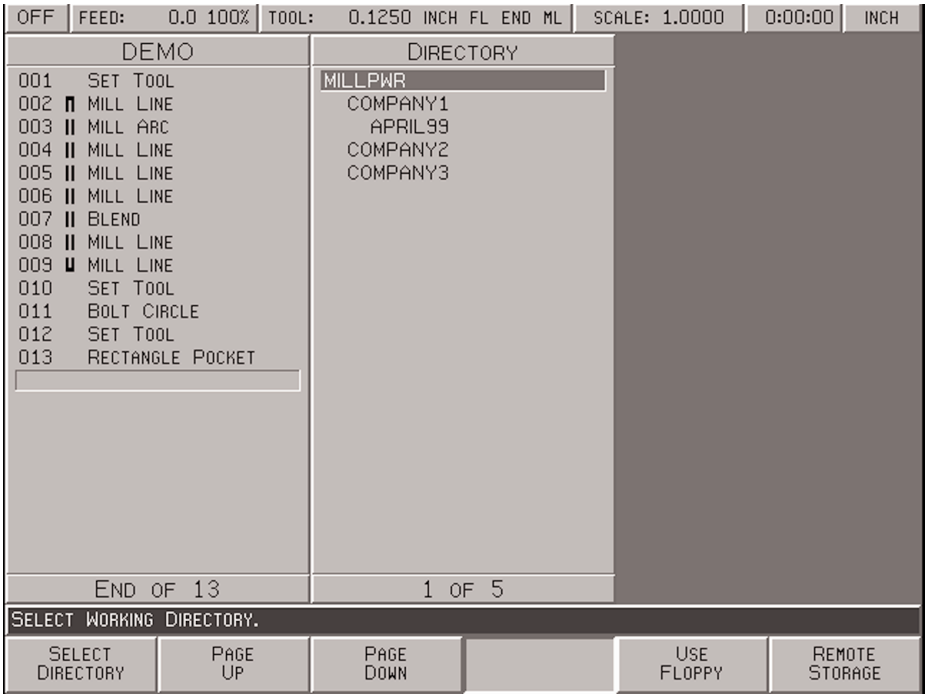
IMPORTANT

Creating a directory does not mean that the directory is selected. If you plan to save your current program in the directory you just created, you must select the new directory first. Otherwise, your program will be saved in the last directory that was selected.

Selecting a Directory

The **SELECT DIRECTORY** softkey allows you to open any of the directories that you've previously created on **MILL*PWR***'s internal memory, on a 3½" floppy disk, or on your PC. You may use this feature any time you save or load a program.

- Press the **PROGRAM FUNCTIONS** softkey, then the **DIRECTORY** softkey. Press the **SELECT DIRECTORY** softkey—a directory list will appear.



- Indicate where the directory you want to select is located.

  On **MILL*PWR***'s *internal memory:*
  "MILLPWR" and any subdirectories that you have created should appear under the "DIRECTORY" heading.

  On a 3½" floppy disk:
  Insert the 3½" floppy disk containing the directory into the floppy disk drive (located in the lower right-hand corner on the front of the operator console) and press the **USE FLOPPY** softkey. "A:" and any subdirectories you have created should appear under the "DIRECTORY" heading.

  On your PC:
  Press the **REMOTE STORAGE** softkey. "REMTSTOR" and any subdirectories you have created will appear under the "DIRECTORY" heading.

The REMOTE STORAGE softkey will only appear if* MILL*PWR has been set up for remote storage.

- Using the arrow keys, highlight the directory you want to open. (If the list is long, use the **PAGE UP** and **PAGE DOWN** softkeys to scroll through the list more quickly.)

- Press the **SELECT DIRECTORY** softkey again. The "DIRECTORY" list will disappear, indicating that your directory has been selected.

You can now save your program in the directory you have chosen; or if you prefer, load an established program from the directory you selected (refer to **Program Functions**).

Deleting a Directory

MILL*PWR* will not delete directories that contain programs. You must delete each program and subdirectory stored within the directory first (refer to **Deleting a Program**).

To delete a directory:

- From the PGM screen, press the **PROGRAM FUNCTIONS** and **DIRECTORY** softkeys, then press the **DELETE DIRECTORY** softkey.

- Identify where the directory you want to delete is located.

  On **MILL*PWR***'*s internal memory:*
  "MILLPWR" and any subdirectories you have created should appear under the "DIRECTORY" heading.

  On a 3½" floppy disk:
  Insert the 3½" floppy disk containing the directory into the disk drive (located in the lower right-hand corner on the front of the operator console) and press the **USE FLOPPY** softkey. "A:" and any subdirectories you have created will appear under the "DIRECTORY" heading.

  On your PC:
  Press the **REMOTE STORAGE** softkey. "REMTSTOR" and any subdirectories you have created will appear under the "DIRECTORY" heading.

  The REMOTE STORAGE softkey will only appear if* MILL-*PWR has been set up for remote storage.

- Using the arrow keys, highlight the directory you want to delete.

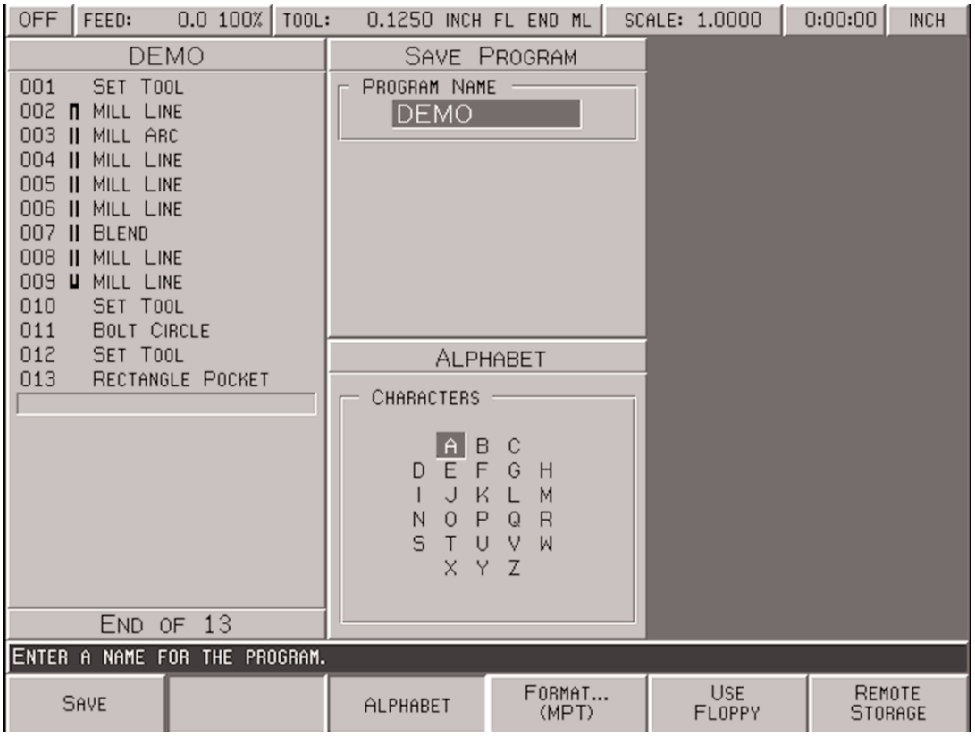
- Press the **ENTER** key. Press the **YES** softkey to erase the directory or the **NO** softkey to cancel.

Saving a Program

You can save your programs in any of three places—on **MILL*PWR***'s internal memory, on a 3½" floppy disk, or on your PC. It is always a good idea to save your programs often to avoid losing valuable information.

- From the PGM screen, press the **PROGRAM FUNCTIONS** softkey, then select the directory where you want to save your program (refer to **Selecting Directories**).

- Return to the PGM screen, then press the **PROGRAM FUNCTIONS** and **SAVE** softkeys. You will be asked to name your program (refer to **Naming a Program**).

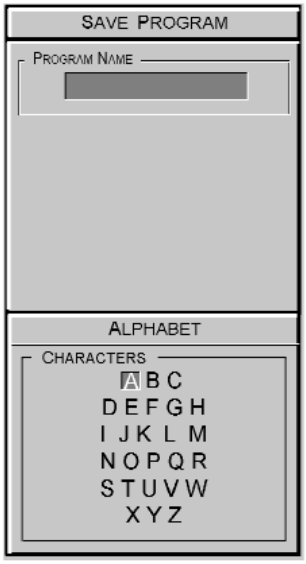


- Press the **SAVE** softkey. The program's name should now appear in the left column above the program steps.

If you make any changes, make sure that you save your program again.

Naming a Program

Before you can save a program, **MILL*PWR*** requires you to name it.



- If you want to use letters, press the **ALPHABET** softkey. An alphabet menu will appear below the "Program Name" field.

- Using the arrow keys, move the cursor from one letter to the next. Press the **ENTER** key to select a letter.

To add numbers to your program name, simply press any of the number keys on the keypad.

You may choose up to eight characters for your program name, mixing numbers and letters if you wish.

- Press the **SAVE** softkey. **MILL*PWR*** will store your program in the directory you have selected.

A message will alert you if the program was not saved properly, or if the name that you've chosen already exists.

If you have accidentally selected the wrong letter or number, press the CLEAR key and rename your program.

Loading a MILL*PWR* (MPT) Program

The **LOAD** softkey allows you to open programs that have already been saved. The steps below tell you how to load a program from **MILL*PWR***'s internal memory, a 3½" floppy disk, or your PC.

If you're loading a program that was created on a two-axes system, you will have to add Z-axis information before you run the program.

If you don't want to add the Z-axis information, consider disabling the Z-axis control and running in 2 axis mode.

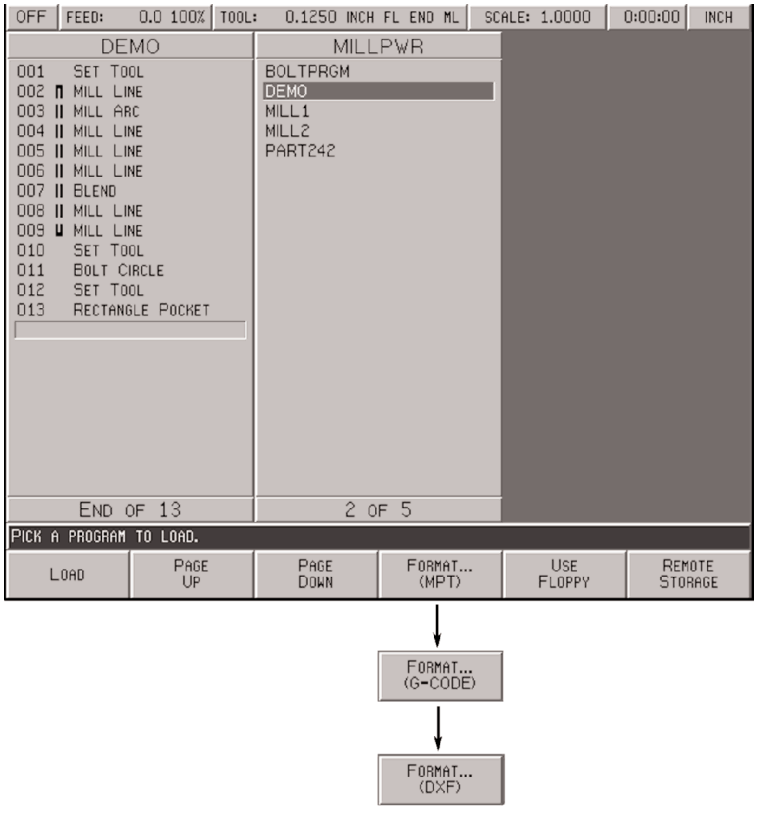
Refer to "Setup" for further instructions.

From MILL*PWR*'s internal memory:

- Save, then clear any open programs.
- From the PGM screen, press the **PROGRAM FUNCTIONS** softkey, then press the **LOAD** softkey. The last directory that was selected and any programs it contains will appear.

If the program is saved in the **MILL*PWR*** *directory:*

- "MILLPWR" should appear at the top. If it doesn't, check that the **USE FLOPPY** and **REMOTE STORAGE** softkeys are not selected.
- Press the format button until MPT is displayed.
- Using the arrow keys, highlight the program you want to load, then press the **LOAD** softkey. The program you've selected should appear on your screen.



If the program is saved in a different directory:

- Press the **CANCEL** softkey.
- Press **PROGRAM FUNCTIONS** softkey.
- Press the **DIRECTORY** softkey.
- Now press the **SELECT DIRECTORY** softkey.
- Using the arrow keys, highlight the directory that contains the program you want to load.
- Press the **SELECT DIRECTORY** softkey again. The "DIRECTORY" screen will disappear. Now that you've selected the appropriate directory, you can load your program.
- Press the **LOAD** softkey. The directory you chose should appear.
- Highlight the program you want to load, then press the **LOAD** softkey. The program you selected should now appear on your screen.

From a 3½" floppy disk:

- Save, then clear, any open programs.
- From the PGM screen, press the **PROGRAM FUNCTIONS** softkey.
- Insert the 3½" floppy disk containing the file you want to load into the floppy disk drive. Located in the lower right-hand corner of the operator console.
- Press the **DIRECTORY** softkey and then press the **SELECT DIRECTORY** softkey.
- Press the **USE FLOPPY** softkey. The “A:” directory and any sub-directories should appear.
- Select the directory that contains your program and then press the **USE** softkey.
- Press the **LOAD** softkey.
- Press the format key until MPT is displayed.
- Using the arrow keys, highlight the program you want to load. Press the **LOAD** softkey. The program you selected should now appear on your screen.

From your PC:

- Save then clear, any open programs.
- From the PGM screen, press the **PROGRAM FUNCTION** softkey, then press the **DIRECTORY** softkey.
- Press the **SELECT DIRECTORY** softkey.
- Press the **REMOTE STORAGE** softkey. (If the **REMOTE STORAGE** softkey does not appear, then it is likely the **MILL*PWR*** and your PC have not been set up. Refer to **Remote Storage** and/or **Installation Setup**.) Select the directory you wish to use.
- Press the **USE** softkey.
- Press the **LOAD** softkey.
- Press the **FORMAT** key until MPT is displayed.
- Select the directory that contains your program (refer to **Selecting a Directory**).

Translating a DXF file:

- Save, then clear, any running programs.
- Locate the directory containing your DXF file (refer to **Selecting a Directory**).
- From the PGM screen, press the **PROGRAM FUNCTIONS** softkey, then press the **LOAD** softkey.
- Select the storage location of your DXF file. (on a 3½" floppy disk or on a PC):

If your DXF file is stored on a 3½" floppy disk:

- Insert the floppy disk containing the DXF file you want to load into the floppy disk drive (located in the lower right-hand corner of the operator console).
- Press the **USE FLOPPY** softkey.
- Press the **FORMAT** softkey until DXF is displayed. The screen will display all of the DXF files stored in the directory you've chosen.
- Using the arrow keys, highlight the DXF file you want to translate.

- Press the **LOAD** softkey.

If your DXF file is stored on a PC:

- Press the **REMOTE STORAGE** softkey. A "REMTSTOR" directory and any programs it contains should appear.

If the **REMOTE STORAGE** softkey does not appear, then it is likely that **MILL*PWR*** and your PC have not been set up properly (refer to **Remote Storage** and **Installation Setup**).

- Press the format key until DXF is displayed. The screen will display all of the DXF files stored in the directory you have chosen.
- Using the arrow keys, highlight the DXF file you want to translate.
- Press the **LOAD** softkey.

MILL*PWR* will read the DXF files you selected, then translate the file's lines, points, arcs, and circles into the appropriate **MILL*PWR*** milling function steps. Default values will be assigned for any information that's missing from the DXF file (such as tool offset, feed rate, etc.).

MILL*PWR* will then arrange the steps in a logical order (based upon common end points) and create a tool path. After the tool path has been determined, the program will appear on your screen.

We suggest that you test your program before machining to ensure that the program steps and tool path do what you want them to do. You can easily edit and rearrange steps as needed (refer to **Step Functions**).

Loading a G-Code file:

- Save, then clear, any running programs.
- From the PGM screen, press the **PROGRAM FUNCTIONS** softkey.
- Select the directory containing your program (refer to **Selecting a Directory**).
- Press the load softkey.
- Press the format softkey until G-Code is displayed.
- Using the arrows, highlight the program you wish to load
- Press the **LOAD** softkey.

If your G-code file is stored on a 3 ½" floppy disk:

- Insert the floppy disk containing the G-code file you want to load into the floppy disk drive (located in the lower right-hand corner of the operator console).
- Press the **USE FLOPPY** softkey.
- Using the arrow keys, highlight the G-code file you wish to load.
- Press the **LOAD** softkey.

If your G-code is stored on a PC:

- Press the **REMOTE STORAGE** softkey. A "REMTSTOR" directory and any programs it contains should appear.

If the **REMOTE STORAGE** softkey does not appear, then it is likely that **MILLPWR** and your PC have not been setup properly (refer to **Remote Storage** and **Installation Setup**).

- Select the directory containing your program (refer to **Selecting a Directory**).
- Press the format key until G-code is displayed softkey. The screen will display all of the G-codes files stored in the directory you've chosen.
- Using the arrow keys, highlight the DXF files you wish to translate.
- Press the **LOAD** softkey.

Note: G-code files are "run only." No editing can be done to the file at the **MILLPWR**. All editing should be done within your CAD/CAM program (refer to **MILLPWR G-code Conventions**).

Running a G-Code Program

Considerations when creating a G-code Program

MILL*PWR* has the ability to read and execute Numerical Code (G-Code) files, however those files can not be edited from the controller. It is important to create and proof the G-code file before attempting to machine a part.

The use of CAD/CAM Software is strongly recommended.

Tool Offsetting

No programmed cutter compensation is used so the tool path should be based on the center and tip of the tool. Do not establish tool length offsets within CAD/CAM software. For repeatable tooling, tool length offsets can be established in the tool library (Diameter offsets are not used).

Using the Tool Library

Each "T" block refers to the corresponding number in the Tool Library. For example, T1 will cause **MILL*PWR*** to retrieve the tool length offset from tool 1 of the Tool Library. **MILL*PWR*** will then offset the spindle by this amount. T2 will cause **MILL*PWR*** to retrieve the tool length offset from tool 2 of the Tool Library, etc. To set up a Tool Library refer to Setup.

It is very important not to have any tool length offsets in the Tool Library if the tooling is not repeatable.

Failure to maintain the Tool Library can cause unpredictable results. Verifying tool length offsets prior to program execution is strongly recommended.

Loading a Program

A G-code program can be loaded into **MILL*PWR*** in the same manner as **MILL*PWR*** programs. (Refer to Loading a Program for instructions on loading a G-code program.) With G-code files, there is no accompanying graphics. Only the program list is displayed. Once loaded, **MILL*PWR*** will indicate lines of code with invalid and unsupported blocks with an "X". If the line containing the code is highlighted, an error message also appears in the message line indicating that the line contains invalid code. The invalid code can be removed by disabling the corresponding functions in the CAD/CAM software and re-posting the G-code program. Simply removing the invalid code from the G-code program can cause unpredictable results and is not recommended.

Running a G-code Program

The tool path should be proofed prior to attempting to machine a part on **MILL*PWR***.

Most CAD/CAM Software packages have this ability. When a program has been proofed and loaded into **MILL*PWR***, lower the knee and dry run the program to verify the tool path and speeds and feeds are correct.

Starting and Stopping a G-code Program
Always start the program from a place in the program where the feed rate, X-, Y-, and Z-axis position are known, such as a tool step. Alternate starting points can be programmed by placing the proper code in the desired locations.

Pressing the **GO** button will cause **MILL*PWR*** to begin executing the current G-code program. Always insure the program step highlighted is an appropriate starting point.

When a program is running, pressing the **STOP** button or the remote pendant will cause the program and all axis motion to pause. Pressing the remote pendant switch again or the **GO** button will cause the program to resume. Pressing the **STOP** button a second time will halt the program execution.

MILL*PWR* - G-Code Conventions

Table of G & M Codes

This table lists both the supported and unsupported codes. Non-supported codes are shown in gray.

Code	Description	Comment
D	Tool Diameter	**MILL*PWR*** does not support automatic cutter compensation. Specifying a tool diameter index while cutter compensation is in effect will generate a run-time error. Specifying a diameter index while cutter compensation is off has no effect and is ignored.
F	Set Feed Rate	The feed rate uses the current program units in effect (ipm or mmpm). The setting applies to current and subsequent blocks. The default is determined from **MILL*PWR*'s** configuration setup.
G0 G1 G2 G3	Linear Interpolation (Rapid) Linear Interpolation (Feed) Circular Interpolation (CW) Circular Interpolation (CCW)	These commands generate table/quill motion. The motion command applies to current and subsequent blocks containing at least one X, Y, or Z coordinate. The default motion command is a linear move at feed (G1).
G4	Dwell	This command causes the system to pause for a specified period of time *prior* to any motion commands within the blocks. The period of time is determined by the P value.
G10	Offset Value Settings	G10 is not supported.
G17 G18 G19	XY Plane Selection XZ Plane Selection YZ Plane Selection	These commands set the plane in which arcs are executed. The setting applies to current and subsequent blocks. The default is G17 (XY).
G20 G21	Set Program Units (Inch) Set Program Units (MM)	These commands set the unit of measure. The setting applies to current and subsequent blocks. The default is G20 (INCH).
G28 G30	Return to Home Reference	**MILL*PWR*** does not have a method for establishing a "home" position. If one or more coordinates are specified in the block, the table/quill will rapidly move to that location. Program execution will continue with the next program block.
G40	Cancel Cutter Compensation	
G41 G42	Cutter Compensation (Left) Cutter Compensation (Right)	**MILL*PWR*** does not support automatic cutter compensation. Enabling compensation (G41 or G42) while a diameter (D is not equal to 0) is in effect will generate a run-time error. Enabling compensation with no tool diameter specified (D = 0) has no effect and is ignored.

Code	Description	Comment
G43 G44 G49	Tool Length Offset (+) Tool Length Offset (-) Cancel Tool Length Offset	**MILL*PWR*** does not support tool length offsetting. The offset is retrieved from **MILL*PWR*'s** tool library when a tool change is executed. These commands are ignored.
G54 to G59	Work Coordinate System	**MILL*PWR*** does not support presettable work coordinate systems. Selecting a coordinate system is possible, but setting it (G10 or G92) will generate a run-time error.
G61 G64	Set "stop" Path Mode Set "continuous" Path Mode	These commands set the path mode. The setting applies to current and subsequent blocks. The default is G64 (continuous).
G80	Cancel Motion Mode	This command cancels the current motion command mode. Motion is reset to G1. X, Y, and Z coordinates are reset to no motion. I, J, and K coordinates are reset to no offset.
G81 to G89	Canned Cycles	**MILL*PWR*** does not support canned drilling cycles (G81) through G89).
G90 G91	Set Offset Mode (ABS) Set Offset Mode (INC)	These commands set the mode for interpreting coordinates. In ABS mode, coordinates are relative to **MILL*PWR*'s** datum. In INC mode, coordinates are relative to the tool's position after completing the previous move. The setting applies to current and subsequent blocks. The default is G90 (ABS).
G92	G92 is not supported.	G92 is not supported.
G94	Set feed/speed units (/sec)	**MILL*PWR*** does not support "per revolution" units. G93 or G95 will generate a run-time error. G 93 & 95 are not supported.
G*		All other G codes not listed will generate a run-time error.
H	Tool Length Offset (Index)	**MILL*PWR*** does not support H-code tool length offsetting. The offset is retrieved from **MILL*PWR*'s** tool library when a tool change is executed. This command has no effect and is ignored.
I J K	X Axis Offset to Arc Center Y Axis Offset to Arc Center Z Axis Offset to Arc Center	The offset applies to current and subsequent arc blocks (G2 and G3). The default is offset.
N	Line Number	Line numbering is optional and for readability only. **MILL*PWR*** does not make use of this information.
O	Program Number	Program number is optional. **MILL*PWR*** does not make use of this information.
M0	Program Stop	This command stops the program after completion of the block. The cursor moves to the subsequent program block. The current settings remain in effect.
M1	Optional Program Stop	This command is ignored. **MILL*PWR*** does not have a switch for selecting whether to stop on this command.

Code	Description	Comment
M2	Program End	This command stops the program after completing the block. The cursor moves to the beginning of the program. The current settings are reset to default values.
M3 M4 M5	Spindle On (CW) Spindle On (CCW) Spindle Off	If spindle control hardware is present, the spindle is turned on or off automatically. If the hardware is not present, the operator is prompted to turn the spindle on or off and/or to set the speed.
M6	Tool Change	If the tool specified is not the current tool (tools do not match), the operator is prompted to manually change the tool. **MILL*PWR*** retrieves the tool length offset from the tool library. If the system does not have repeatable tooling, the operator must establish the Z-axis datum for the new tool before continuing.
M7 M8 M9	Coolant On (Mist) Coolant On (Flood) Coolant Off	If the AMI hardware is present, the coolant is turned on or off automatically. If the hardware is not present, the operator is prompted to turn the coolant on (mist), on (flood), or off.
M30	Program End w / Pallet Shuttle	**MILL*PWR*** does not support control of a pallet changer. This code has the same effect as M2.
M48 M49	Enable Speed/Feed Override Disable Speed/Feed Override	It is not possible to disable feed rate override on **MILL*PWR***. These commands are ignored.
M60	Program Stop w / Pallet Shuttle	**MILL*PWR*** does not support control of a pallet changer. This code has the same effect as M0.
M*		All other M codes not listed will generate a run-time error.
P	Dwell Time	The dwell time is specified in seconds. The setting applies to current and subsequent dwell blocks (G4). The default is 0.0 sec.
S	Spindle Speed	The spindle speed is set to the specified speed (rpm). If automatic spindle control hardware is present, the spindle speed is set immediately. If not present or not enabled, the user is prompted to set the speed. If the spindle is currently off (M5), it will not be turned on unless accompanied by a spindle direction block (M3 or M4). The setting applies to current and subsequent spindle direction blocks. The default is 0 rpm.
T	Tool Selection	The tool selection represents the number of the tool to use at the next tool change. The selection applies to current and subsequent blocks containing a tool change (M6). The default is no tool selection.
X Y Z	X Axis Coordinate Y Axis Coordinate Z Axis Coordinate	The coordinates represent the destination for the G0, G1, G2, or G3 command currently in effect. They use the current units (G20 and G21) and offset mode (G90 and G91).

Additional G-code Conventions for MILL*PWR*

The following lists some of the expectations and limitations of programs imported into **MILL*PWR***.

- Blocks may contain multiple commands and are executed with the following precedence:

  Messages
  Tool Change
  Spindle Control
  Coolant Control
  Dwell
  Motion
  Stop

- Operator comments should be enclosed in parentheses.
- An operator comment with "MSG" appearing within the text is considered a message. The text following "MSG" (up to 22 characters) is displayed to the operator at run-time. Program execution pauses until the operator acknowledges the message.
- Arcs may only be programmed by endpoint and center. Using endpoint and radius will generate a run-time error.
- Parametric programming (use of variables or algebraic operations) is not supported. The characters (#, [,], <, or >) within a block will generate a run-time error.
- Program delimiters ("%") are ignored. Text following the delimiter is ignored.
- Text following a programming comment character (";") is ignored.
- White space is ignored between parameters but not within a numeric value or message.
- If a coolant command (M7, M8 and M9) appears in the block, the Operator Intervention Message is displayed regardless of the current coolant setting. If AMI hardware is present, the block will execute without the need for any operator intervention or acknowledgement.
- All tool diameter compensation must be made in CAD/CAM, however, the tool length off set is read from **MILL*PWR***'s tool library.

  For example: In a G-code file, T1 will use the tool length offset from Tool #1 in the tool library. T2 will use tool length offset from tool library tool #2, etc.

- The skew feature does not work with G-code programs. Remove any skew angle prior to running a G-code program.

Merging Programs

The **MERGE** softkey allows you to merge two **MILL*PWR*** programs together. With this function, all of the program steps within the program you selected will be copied into your current (or open) program. Keep in mind that you can edit any of the new steps if you need to (refer to **Step Functions**).

To merge programs:

It's always a good idea to save your work before merging programs so that you can easily recover your original program if you need to.

- From the PGM screen, highlight the step in your current (open) program where you want the new steps to be inserted.
- Press the **PROGRAM FUNCTIONS** softkey.
- Press the **MERGE** softkey. The last directory that was opened will appear. You can change directories if you need to (refer to **Selecting a Directory**).
- Using the arrow keys, highlight the program that you want to merge into your program.
- Press the **MERGE** softkey again. The steps from the program you highlighted should now appear in your program.

Note: The merge function does not work with G-code files or DXF files.

Backing Up a Program

With the **BACKUP** softkey, you can make backup copies of programs that you have already saved on **MILL*PWR***'s internal memory. You should keep backup copies on hand in case a program is accidentally deleted or modified, or you can't recover the original programs for any other reason.

It's best to save the final version of a program before creating a backup copy. Otherwise, you'll have to back up the program again after you've made any changes.

To back up a program:

- From the PGM screen, press the **PROGRAM FUNCTIONS** softkey, then select the directory containing the program(s) you want to back up (refer to **Selecting a Directory**).
- Now press the **BACKUP** softkey. The following softkey options should appear:

- Indicate where you want to back up your program(s): onto a 3½" floppy disk (press the **USE FLOPPY** softkey) or onto your PC (press the **REMOTE STORAGE** softkey).
- Determine which program(s) you want to back up.

  To back up all of your programs in the directory:

  - Press the **ALL PROGRAMS** softkey. **MILL*PWR*** will store a backup copy of each program in the directory you've selected.

  To back up only one or just a few programs:

  - Press the **SELECT PROGRAMS** softkey.
  - Highlight each program you want to back up press the **SELECT PROGRAM** softkey again. An arrow will appear beside each program name you've selected.

- Now press the **BACKUP PROGRAMS** softkey. **MILL*PWR*** will highlight each program as a backup copy is saved in the directory you've selected.

Note: If a program with the same name is already stored in the directory you've chosen, **MILL*PWR*** will ask you if you want to replace the old copy with the latest copy. Choose the **YES** softkey or **YES TO ALL** softkey to continue or the **NO** softkey to cancel.

Deleting a Program

You can delete any program that has been saved.

To delete a program:

- Select the directory that contains the program you want to delete (refer to **Selecting a Directory**).
- From the PGM screen, press the **PROGRAM FUNCTIONS** softkey, then press the **DELETE** softkey.

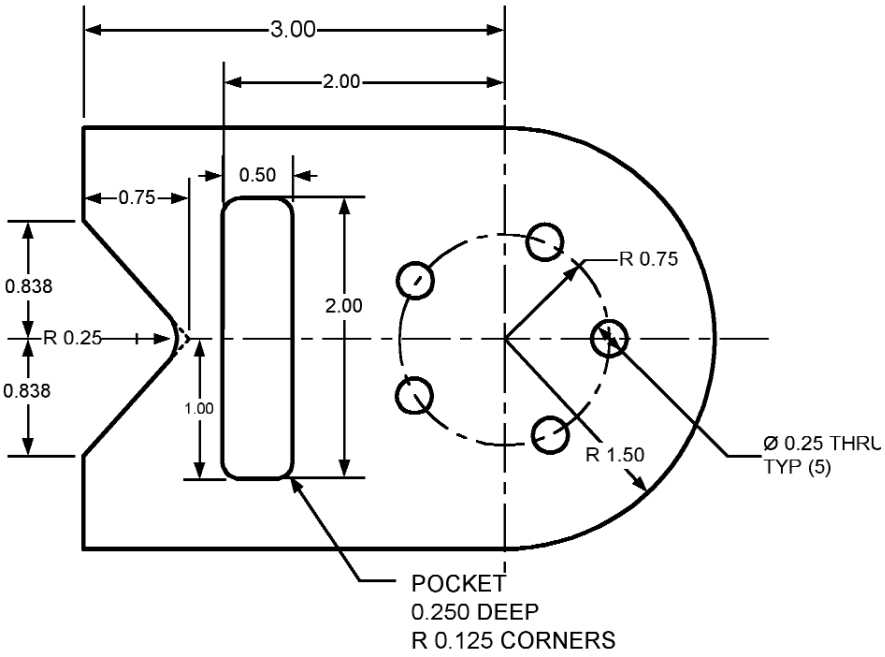
- Using the arrow keys, highlight the program you want to delete.
- Press the **DELETE** softkey. You will be asked if you are sure that you want to delete the program. Press the **YES** softkey to continue or the **NO** softkey to cancel.

IMPORTANT
By answering "Yes," you will erase the highlighted program from memory. Deleted programs cannot be recovered unless a backup file was previously created.

This page intentionally left blank.

Demonstration Program

The following steps and key stroke sequences will guide you through creating a demonstration program based upon information from the print below. This will help introduce you to **MILL*PWR*** and familiarize you with how it operates.



Selecting Datum

Although there is no clear "zero point" identified on this print, use the center of the bolthole pattern as datum. There are several advantages to using this location:

- it is the center of the bolt hole pattern
- it is the center of the large arc, making it easy to calculate the arc's start and end points
- the pocket is dimensioned from this point
- nearly all of the dimensions on the print originate from this point

This print could also be programmed using another point—the upper left corner, for example—as datum.

The datum will be ready to set just before running a program.

Beginning The Program

Begin by pressing the **PGM** key.

Selecting A Tool

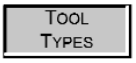
The logical first step for most programs is to choose the tool that you want to begin with. Let's use a 1/4" **FLAT END MILL**.

- Press the **TOOL** key.

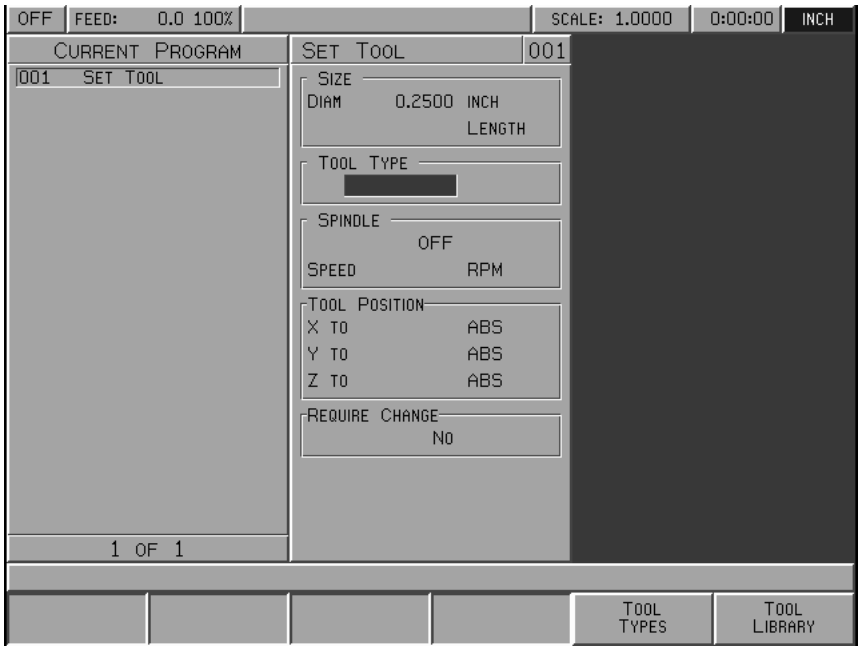
- Enter a value of 0.25
  The tool length is optional— leave it blank.

- Arrow down to **TOOL TYPE.**

- Press the **TOOL TYPES** softkey.



- Arrow down to **FLAT END MILL** and press the **ENTER** key.



Consider entering a Tool Position. This will enable you to go to a location away from your workpiece to change tools.

- Arrow down to **SPINDLE**.

- To use a forward spindle direction. Press the **FORWARD** softkey then press the **ENTER** key.

- Set the cutter's spindle speed for 1300 RPM.

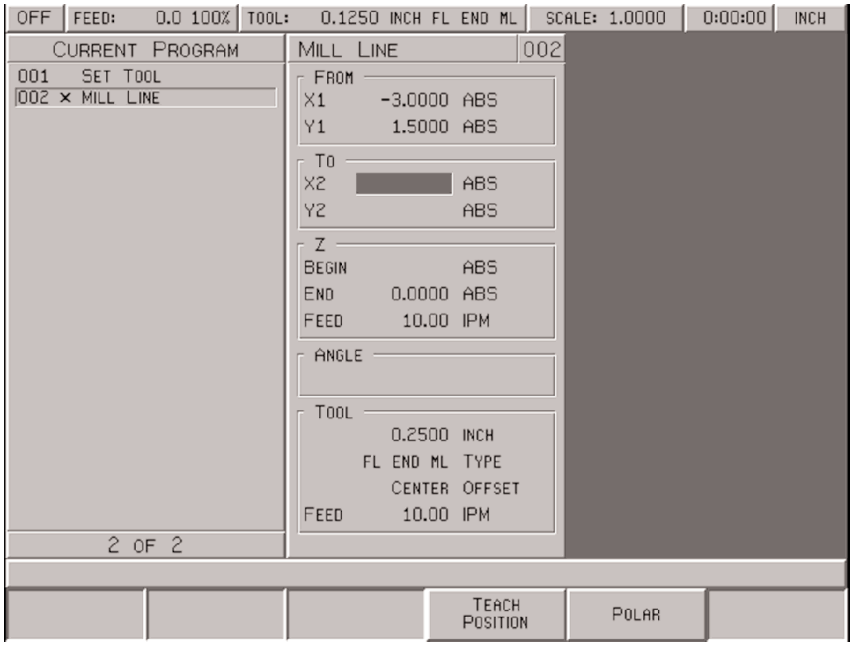
- Press the **USE** key.

You don't have to press the **ENTER** key after the last value—you can just press **USE**.

Programming the Contour

This part can begin at several different places. Begin at the upper-left corner, and cut in a clockwise (CW) direction.

- Press the **LINE** key.



To enter a negative number, use the "±" key, not the "-" or "+" key. The "-" and "+" keys are for performing math operations within a numeric field.

- Using the keypad, enter the following information:

FROM: X1 = -3 ABS

Y1 = 1.5 ABS

TO: X2 = 0 ABS

Y2 = 1.5 ABS

Z BEGIN DEPTH: .02 ABS

Z END DEPTH: -.25 ABS

- Now highlight the “Offset” field.

  The tool specifications will be filled in automatically from the information we entered in step 001.

- To cut around the outside of the contour in a clockwise direction, use a left offset. Press the **LEFT** softkey.

  MILL*PWR* has been factory set with a feed rate of 10 inches per minute, which is fine for this operation.

- Press the **USE** key.

  Notice that the line is immediately displayed on the Program (PGM) screen. This is called immediate-part-view graphics.

Next program the arc:

- Press the **ARC** key.

MILLPWR assumes that you are continuing from where you left off so it automatically fills in the "From" point, the depth and tool information for you.

- Enter the following information into the "To" and "Radius" fields:

TO: X2 = 0 ABS

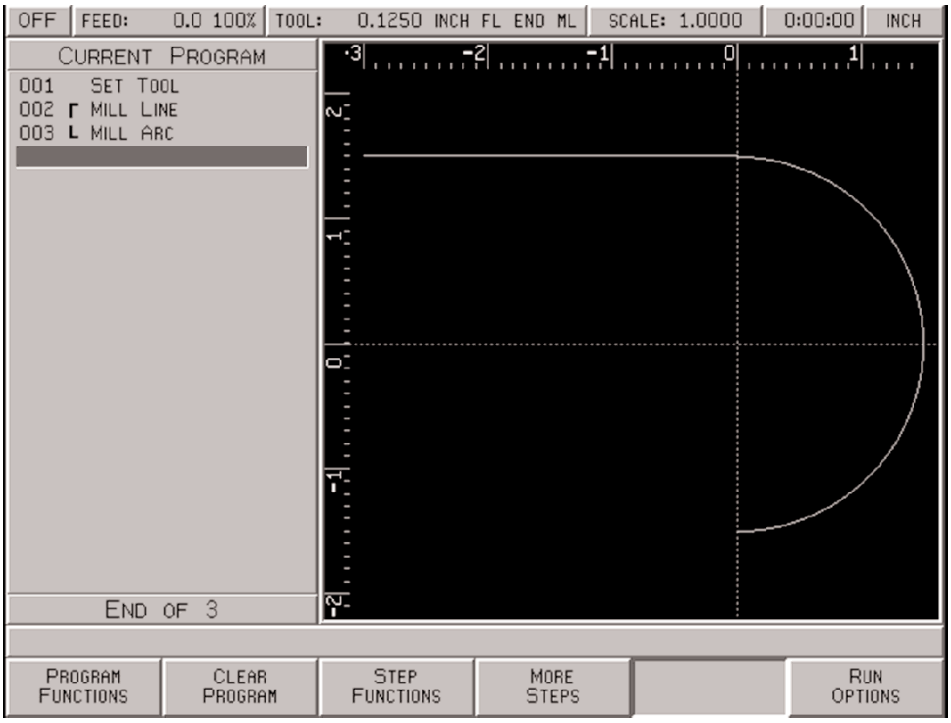
Y2 = -1.5 ABS

RADIUS: 1.5 MINOR

Since this arc will be starting at the top and moving around to the bottom, choose **CW** for the direction.

DIRECTION: CW

- Press the **USE** key.



Notice the lines connecting steps 002 and 003 in the program list. This indicates that the line and arc form a continuous contour.* MILL*PWR* *will cut them without stopping.

Next, enter the line that starts at the bottom of the arc.

- Press the **LINE** key and enter the following information:

  TO: X2 = -3.0 ABS

  Y2 = -1.5 ABS

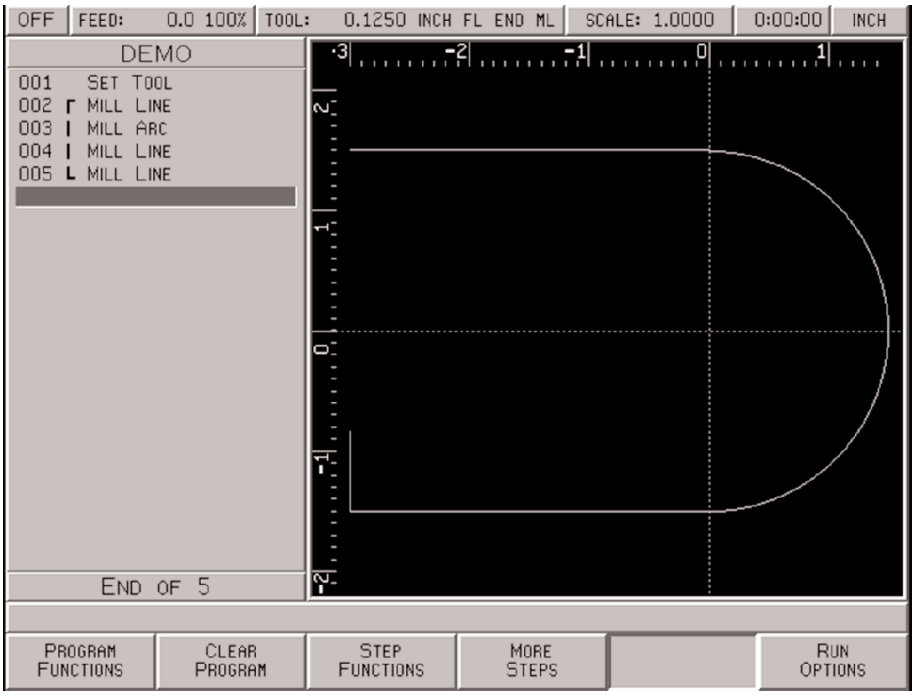
- Press the **USE** key.

- Press the **LINE** key to add another line. Now enter the following information:

  TO: X2 = -3.0 ABS

  Y2 = -0.838 ABS

- Press the **USE** key.



- Press the **LINE** key.

- Enter the following information:

  TO: X2 = .75 INC

- Now press the **ABS/INCR** key.

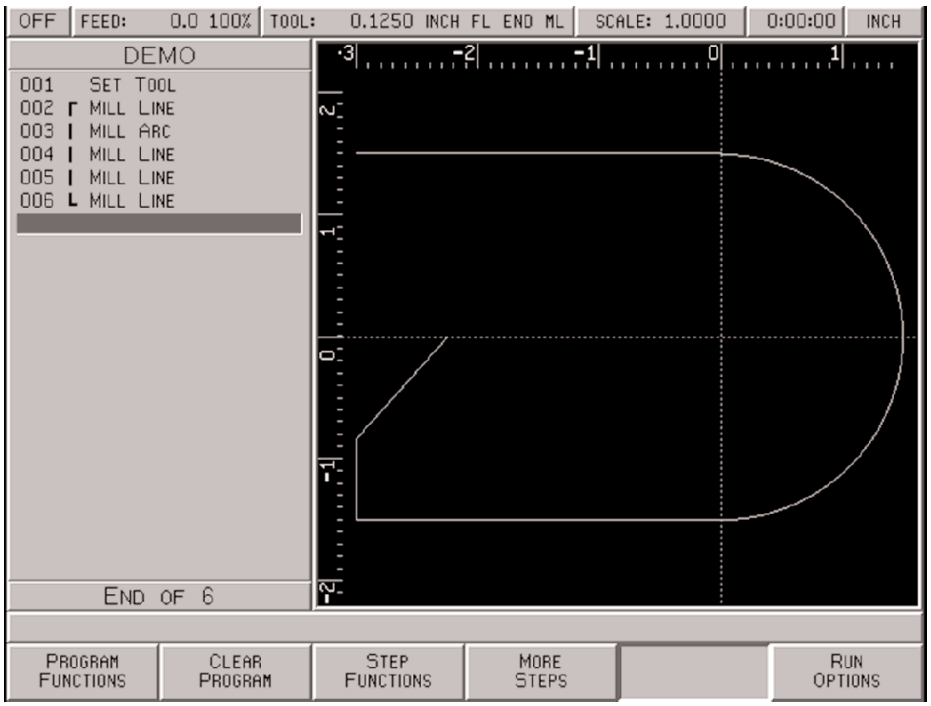
MILL*PWR* assumes that you want to use the X coordinate of the "From" point (X1) as your incremental reference, which is exactly what is needed.

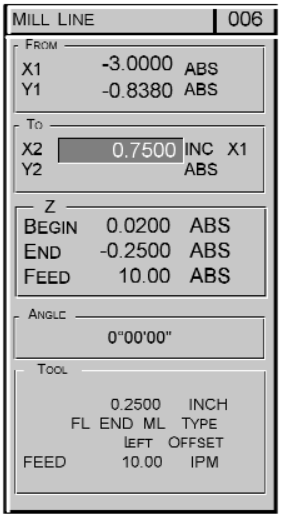
- Press **ENTER** to confirm this and again to accept the value.

- Now enter the location of Y2 in the "To" field:

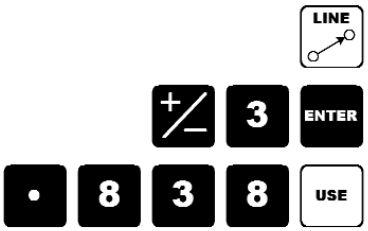
  TO: Y2 = 0 ABS

- Press the **USE** key.





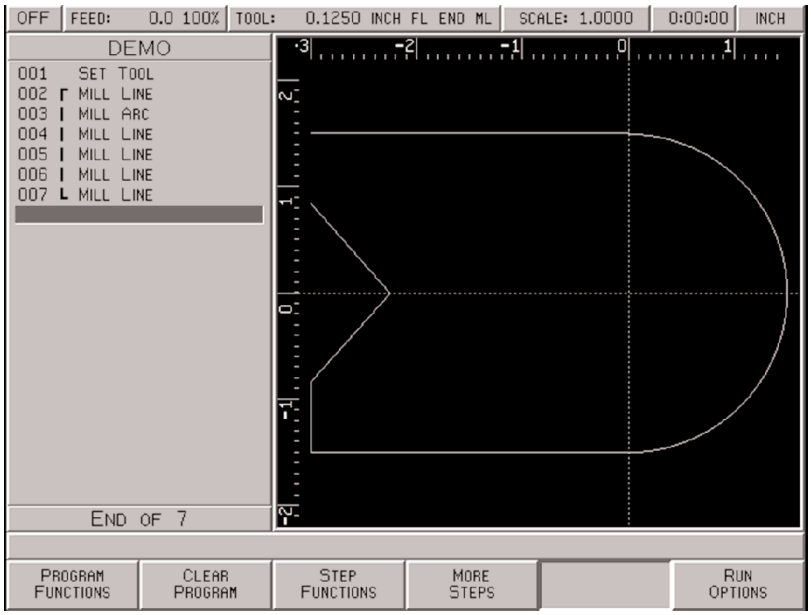
- Press the **LINE** key again.



TO: X2 = -3.000 ABS

Y2 = 0.838 ABS

- Press the **USE** key.



Now go back and insert a **BLEND** between steps 006 and 007.

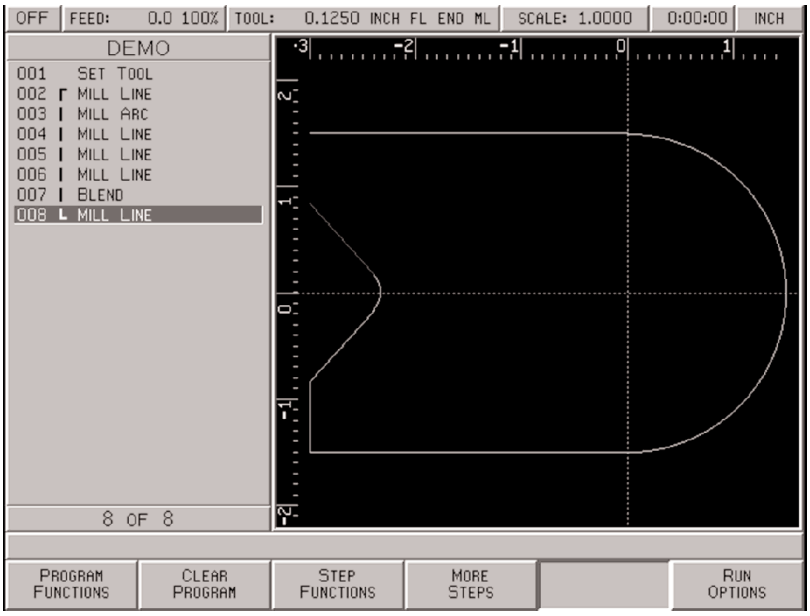
- Using the **UP** arrow key, highlight the last step, 007 Mill Line.

- Press the **BLEND** key.

You can see in the program list that step 007 has moved to step 008 and that the blend will be inserted into step 007.

Notice the “Steps” field is indicating that steps 6 and 8 will be blended.

- Enter a radius of 0.25 and press the **USE** key.



Notice how the last two lines are now "blended" together with a radius.

The **BLEND** step could just as easily have been inserted immediately after line 006. In doing so, the **BLEND** step *would* show up in the listing, but it *would not* be displayed graphically until the step that follows is added.

- Press the down arrow key once to reach the end of the program. Now more steps can be entered.

- Press the **LINE** key.

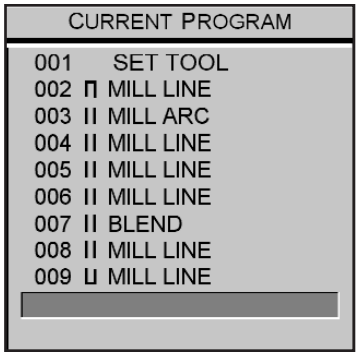
TO: X2 = -3.0 ABS

Y2 = 1.5 ABS

- Press the **USE** key.

Notice how the lines next to the program steps have changed. They now indicate that the contour is closed.

This happens when the “To” point of the last step is the same as the “From” point of the first step.

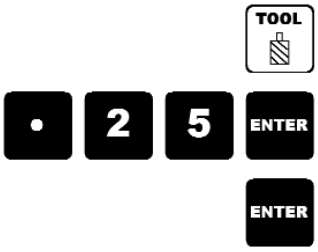


Programming the Bolthole Pattern

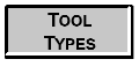
Begin by changing the tool.

- Press the **TOOL** key and enter 0.25" for the diameter.

  Again, skip the tool length.



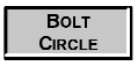
- Press the **TOOL TYPE** softkey.



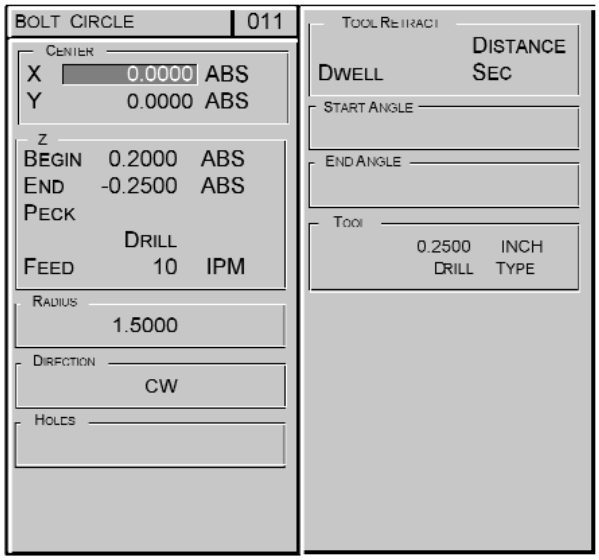
- In the “Tool Type” field, arrow down to “Drill” and press **ENTER**.

- Adjust the spindle direction and speed as needed.
- Press the **USE** key.

- Press the **HOLES** key and then press the **BOLT CIRCLE** softkey.



The following screen will appear:



Notice that the **CENTER**, **DIRECTION** and **Z BEGIN** are carried forward from the arc that were programmed earlier. Enter the depth of cut, peck radius and tool retract values.

- Arrow down to the “Z End” field and enter a value of -0.4

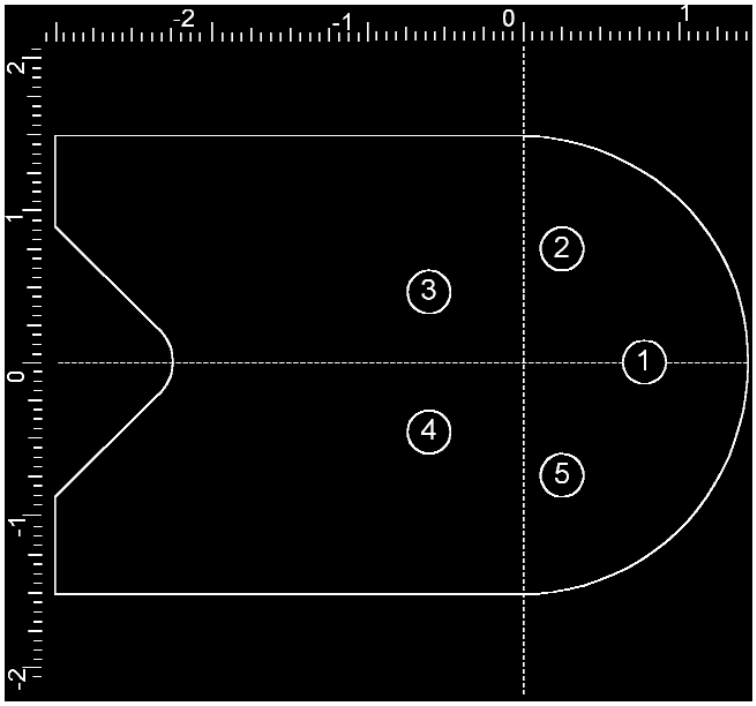
- Enter 6 in the “Peck” field.

- Arrow down again and enter a radius of 0.75”

- Press the “down” arrow key again, then enter 5 for the number of holes.

- Highlight the “Number” field in tool retract then enter a value of 3.
- In the “Dwell” field, enter 5 seconds as the length of time needed for the quill to pause during the retract cycle.
- Press the **USE** key.

In this example, the starting and ending angles were not changed. As a result, your first hole will be placed at zero degrees. If you were to look at the face of a clock, zero degrees will be at 3 o’clock. Without a specified ending angle **MILL*PWR*** will space the number of holes entered evenly around a full circle. If a counter-clockwise direction was applied to the hole pattern, the second hole of our five hole pattern will be between 12 o’clock and 1 o’clock. The holes continue around the circle as shown below.

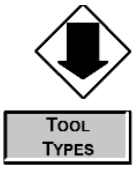


Programming the Rectangular Pocket

First, let's enter the tool that is needed to machine the pocket.

- Press the **TOOL** key and enter the data for a 0.125" diameter flat end mill.

- Arrow down and highlight the "Tool Type" field.
- Press the **TOOL TYPE** softkey.

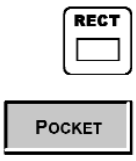


- Highlight "Flat End Mill."

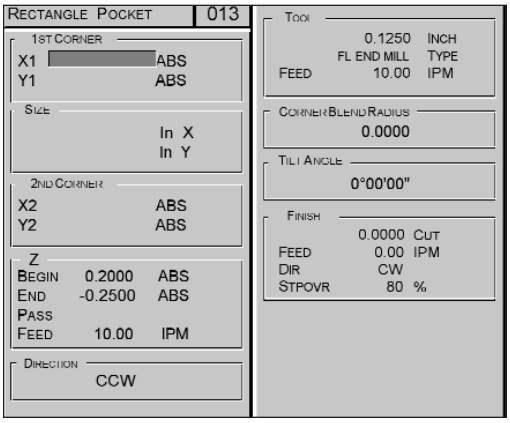
- Press the **ENTER** key.

- Adjust the spindle direction and speed.
- Press the **USE** key.

- Press the **RECT** (rectangle) key, then press the **POCKET** softkey.



The “Rectangle Pocket” screen will appear:



- Enter the following information:

1ST CORNER: X = -2.0 ABS

Y = -1.0 ABS

SIZE: 0.5 ABS in X

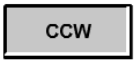
2.0 ABS in Y

- Arrow down and highlight the “Pass” field.

- Enter 2 for the number of passes.

- Arrow down and enter the following information:

DIRECTION: CCW



- Arrow down and enter the following information:

CORNER BLEND RADIUS: 0.125

- Press the **USE** key.

Saving Your Program

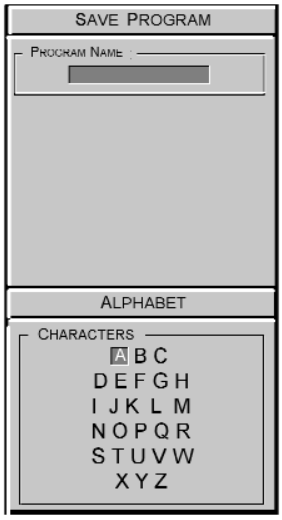
Your demonstration part program is now complete.

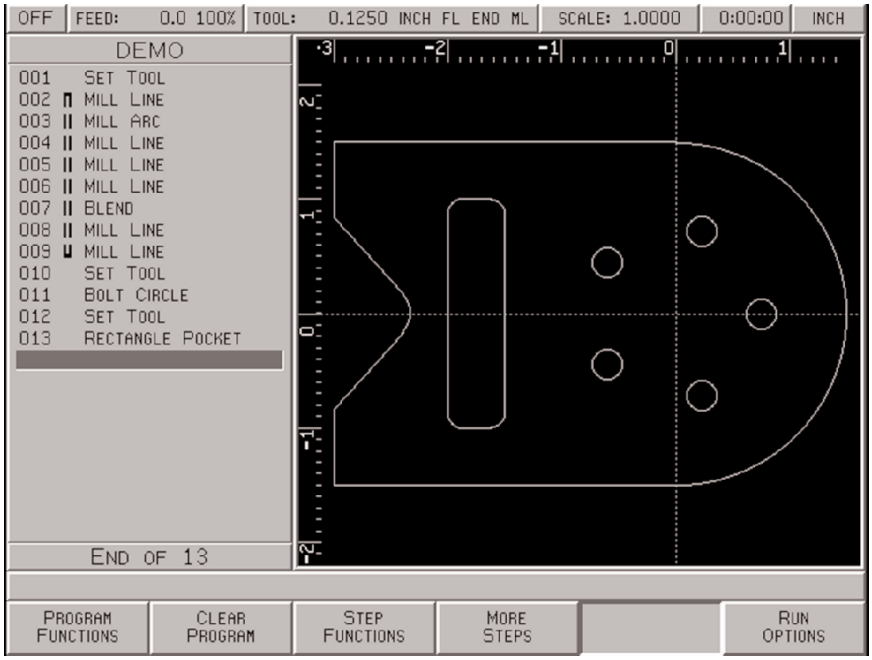
- To save your program, press the **PROGRAM FUNCTIONS** softkey.
- Press the **SAVE** softkey.
- You can name your program by pressing the numeric keys or by pressing the **ALPHABET** softkey, highlighting a letter, and then pressing the **ENTER** key. You may select up to eight characters, mixing numbers and letters if you wish.
- After you've named your program, press the **SAVE** softkey to save it.

You will be warned if a program already exists with the name that you have entered.

Saving the program means that it is stored and will not be lost if there is a power interruption.

You can retrieve the program later by pressing the **PROGRAM FUNCTIONS** and **LOAD** softkeys.



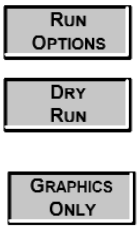


Testing Your Program

It's always a good idea to test your program before you cut your part.

Dry Run with Graphics Only

- Press **RUN OPTIONS** and then select both the **DRY RUN** and **GRAPHICS ONLY** softkeys.



- Press **RUN OPTIONS** again to finish.

- Move to the start of the program by pressing the **1** key followed by the **ENTER** key.

- Now press the **GO** key. Watch as **MILLPWR** shows you how the part will be cut at dry run speed.

Dry Run with Movement

- Press **RUN OPTIONS**. Press **GRAPHICS ONLY** to de-select it.

⚠ Before you press the **GO** key to begin the dry run, ensure that the tool will not touch the workpiece when the quill begins to move. To avoid tool interference, we suggest that you do one or more of the following:

- Lower the knee
- Remove the tool or workpiece
- Reestablish datum away from the part (refer to **Establishing Datum** located in Section 2).

If your cursor is positioned past the last step, **MILL*PWR*** *will automatically go to step 001 when you press GO.*

- Now press the **GO** key and check that the Z-axis is engaged.

- Press the **GO** key again. **MILL*PWR*** will run each step at dry run speed without stopping.

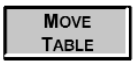
If the part falls entirely within the workpiece, you can cut the part.

Running the Program

The first step in running a new program is to establish datum. Remember to chose the center of the bolt circle as datum.

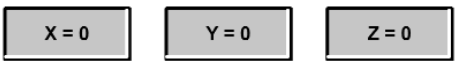
- Place your workpiece into a vise.
- Place the tool you want to use first into the spindle.
- Using the **MOVE TABLE** softkey and the arrows, position the tool over the workpiece where you want the center of the bolt circle's datum to be.

- After you've finished moving the table, press the **MOVE TABLE** softkey again to de-select it.



- Touch the tool to the top surface of your workpiece.
- From the DRO screen, press the **DATUM** softkey.

- Press the **X = 0**, **Y = 0** and **Z =0** softkeys to establish the current tool position as datum.



- Press the **ARROW DOWN** softkey.

- Set the Z retract position 1" above the workpiece.

- Press the **USE** key.

- Return to your program by pressing the **PGM** key. Perform a second dry run, this time allowing the table to move, to see if there is enough material to run the program. Set zero.

Tool Changes

Whenever you encounter a **SET TOOL** step, **MILL*PWR*** will display the DRO screen and let you know which tool to load. If no tool diameter was programmed in the "Tool Step," you will be prompted to provide one.

- Use the **MOVE TABLE** softkey and arrow keys to move the tool away from your workpiece.

- Insert the required tool into the spindle.
- Using the **MOVE TABLE** feature, position the tool over the surface of a known depth on your workpiece.

- Press the **DATUM** softkey.

- Position your tool so that it touches a known surface, then enter that position into the Z-axis.

If the surface is at datum 0, use the Z = 0 softkey.

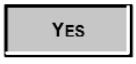
- Press the **USE** key.

- Raise the tool and press the **GO** key to continue the program.

Clearing the Program

- From the PGM screen, press the **CLEAR** softkey.

- Press the **YES** softkey. The PGM screen will be cleared and ready for another program.



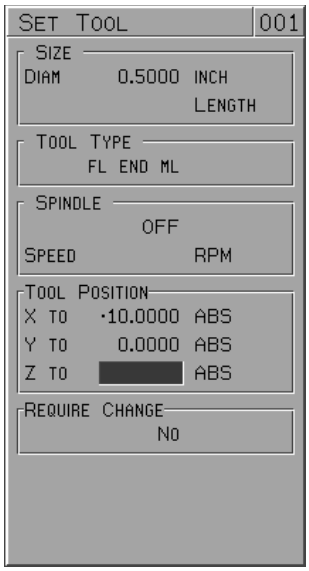
Simple Milling & Drilling

Most of the program steps described in this section can be performed as one-step milling functions from the DRO screen or included as steps in a program (press the **PGM** key).

Set Tool

"Set Tool" defines your tool and should appear as the first step in all of your programs. You should also insert a "Set Tool" step anywhere you want to change tools. **MILL*PWR*** will apply your latest tool setting to the program steps that follow.

To program a “Set Tool” step:



- Press the **TOOL** key.
- Enter the tool's diameter.

Consider entering a Position/Drill step before the Set Tool step. This will enable you to go to a location away from your work-piece to change tools.

Note: If you leave the “Diam” field blank, you will be prompted to provide a diameter once the program is running.

Note: Tool length offsets are mentioned later on within this section.

- Highlight the "Tool Type" field.
- Press the **TOOL TYPES** softkey for a list of available tool types.
- Highlight a tool type, then press the **ENTER** key.

If you press the TOOL LIBRARY softkey, highlight a tool and press ENTER, **MILL*PWR*** *will fill in the "Diam," "Length" and "Tool Type" fields for you.*

- For spindle control, select from the **OFF**, **FORWARD** and **REVERSE** softkeys.

  If you selected "Forward" or “Reverse,” enter a spindle speed. An Operator Intervention Message (OIM) will prompt you to set the speed and direction when you run your program. (If your machine's set up to control the spindle, **MILL*PWR*** will adjust the speed and direction for you.)

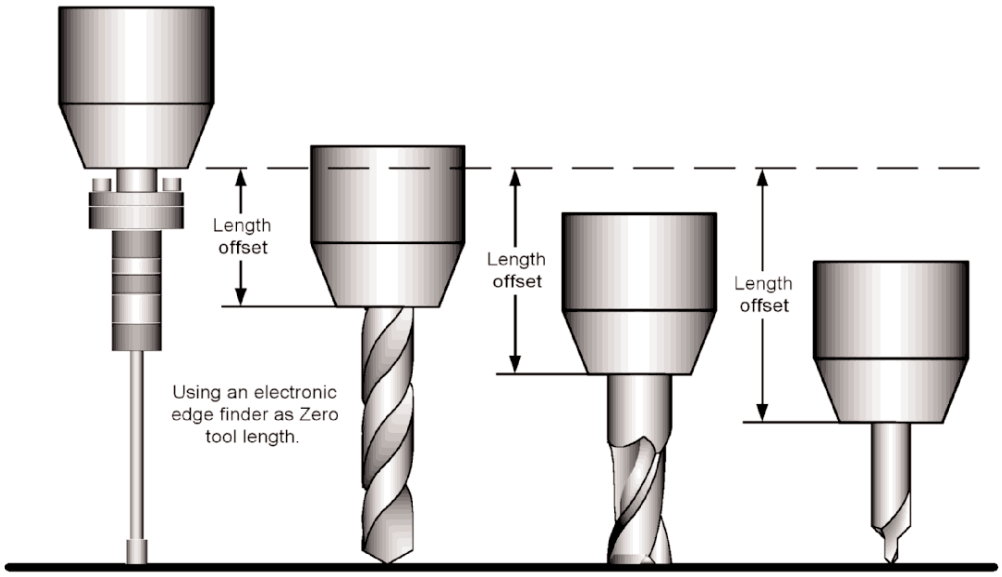
- Tool Position allows the operator to enter a position to which the tool will move so you can perform a tool change while running a program.

- Required Change is used if the operator, for any reason, wants to move the tool into position for a tool change even if there is no tool change actually required. Pressing **YES** for “Required Change” field will activate this prompt.

- Press the **USE** key.

Programming a Tool Step with Repeatable Tool Length Offsets

See the graphic below to identify the tools you will be using.



If you begin setting the tool length offsets by setting Datum using an electronic edge finder it must have a fixed and repeatable length. The current tool information cannot have a length value. All of the tool length offsets in the program will be the difference in length between the tool and the edge finder.

When defining tool length offsets in the Tool Step, touch each tool to the same surface and press the **TEACH TOOL LENGTH** softkey to establish the tool length.

Once you have identified the tools you will be using, you can program the length offsets into the tool steps.

Entering the first tool

Set your Z-axis Datum with a tool or standard using no tool length offset. In our example, we use the first tool in the program to set datum zero. If you wish to use a 3-D electronic edge finder or tool holder with a repeatable standard, then place it into the spindle instead, and set datum. Then you find the tool length offset for tool 1 as well.

- From the **DRO** screen, press the **TOOL** key.
- Enter the diameter and then enter zero for the tool length.
- Press **GO**.
- An Operator intervention message will appear asking you to "use" the entered tool.
- Place your first tool in the spindle and press **GO** again.
- **MILLPWR** is now set with a zero tool length offset.
- Press the **DATUM** softkey.
- Touch the current tool to the top of the workpiece.
- Press the "**Z=0**" softkey, then the **USE** key.

Use this same location to "teach" each tool length in the program.

- Press the **PROGRAM** key and highlight the first tool step.
- Press **ENTER**.
- Use the numeric keypad to enter the tool's diameter.
- Highlight the length field and insure the length offset is zero. (If you used a 3-D electronic edge finder or tool holder with a repeatable standard to set datum, then touch tool 1 to the same surface you used to set the Z-axis datum above and teach its tool length here.)
- Highlight the next tool step and press **ENTER**.
- Use the numeric keypad to enter the tool's diameter.
- Place this tool into the spindle and touch its tip to the same surface you used to set the Z-axis datum Zero above.
- Press the **TEACH TOOL LENGTH** softkey. The length offset for this tool will appear in the length field.
- Highlight the "unit" field. If the diameter you entered is metric, press the **MM** softkey.
- Highlight the "Type" field.
- Press the **TOOL TYPES** softkey.
- Highlight the type of your tool from the list of available tool types and press **ENTER**.
- Press **USE**, and repeat this procedure for each additional tool in the program.
- If your Z-axis datum is not located at the top of your workpiece, you will need to reset your Z-axis datum. Be sure to set your Z-axis datum using the tool shown in the "current tool" box located in the status bar on the top of the screen. **If the tool doesn't match, the datum will not be correct.**

After the tool information is established in the program, later you can edit the values as needed to adjust for accuracy and wear.

Changing to a Tool of unknown length when in the DRO

If you are using the DRO and need to set a new tool, follow the procedure below. Be sure to reestablish the tool length offset for your program before running it as described below.

- From the **DRO** view, press the **TOOL** key.
- Enter the tool diameter, clear any length and if you wish, the type.
- Press the **GO** key.
- An Operator intervention message will appear asking you to "use" the selected tool.
- Place the selected tool in the spindle.
- Press the **GO** key to confirm that you have installed the tool.
- Set the Z-axis datum following the instructions found in your operation manual.

At this point, your program's tool length offsets are no longer valid. Follow the steps below to reestablish the Z-axis datum and tool length offsets for your program.

- From the **PROGRAM** view, highlight any tool step.
- Press the **GO** key.
- An Operator intervention message will appear asking you to "use" the programmed tool.
- Place this tool in the spindle.
- Press the **DATUM** softkey.
- Reset the Z-axis datum following the instructions found in your operation manual. By resetting Datum during a tool step within the program, all of the tool offsets in the program become valid again.

Changing to a Tool of unknown length in program

When running a program, it is easy to switch between tools of known length, to tools that have a non-repeatable length such as a counter-bore held in an R-8 collet. For example, let's say a program is being run that has tool 1 -a center drill- in a repeatable holder, as the first step.

Step 2 in the program is a bolthole pattern.
Step 3 is a tool step using tool two-a drill- in a chuck that also repeats.
Step 4 is the same hole pattern repeated.
Step 5 is a tool step programmed as a counter bore of unknown length.
Step 6 is the same hole pattern repeated.

For this example, if two identical parts are being machined. The first part is in place and datum has been set using tool one from the program as the current tool.

The top of the workpiece is Z-zero. When the **GO** button is pressed, **MILL*PWR*** will see that step one is asking for the same tool as the current tool and will proceed on the next step. When the next tool step is executed, you will be prompted to change tools to the drill. Simply change the tool and press **GO** to continue. The next tool-step 5- is of unknown length. When this step is executed, and you are prompted to change tools,

- Press the **DATUM** softkey. The **DRO** screen will appear with the set datum window.
- Place the counter bore in the spindle and touch the cutting edge to the top of the workpiece.
- Press the **Z=0** softkey and then **ENTER**. The tip of the tool is now set at datum zero.
- Press **GO** to continue with the program.

Since the Z-axis datum was set using a tool of unknown length, the programmed tool lengths will not repeat until the Z-axis datum is reset again, using a tool of known length. This can be done when step 1 is run on the second part. After the part has been changed,

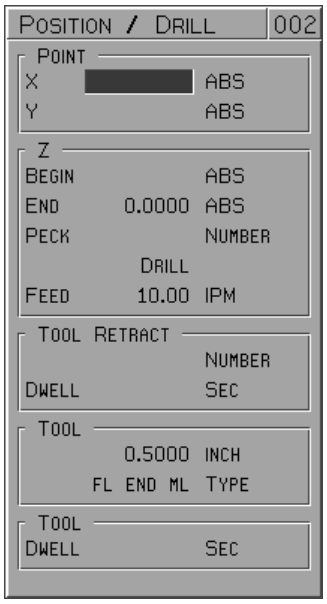
- Press **GO**. You will be prompted to change tools to the center drill.
- Press the **DATUM** softkey. The **DRO** screen will appear with the set datum window.
- Place the center drill in the spindle and touch the tip to the top of the workpiece.
- Press the **Z=0** softkey and then **ENTER**. The tip of this tool is now set at datum zero.
- Press **GO** to continue with the program.

Now that datum has been reset with a tool of known length, all of the preset tools again relate to datum. The program can continue as usual.

Position/Drill

POS

The position/drill function will move your table to the position you want based upon your X- and Y-axes coordinates.



To program a position/drill step:

- Press the **POS** key.
- Enter the X- and Y-axes coordinates.
- Enter the begin and end depths for the Z-axis.
- (Optional) Enter either the number of pecks OR the distance between each peck.
- Select the **DRILL**, **BORE** or **POSITION** softkey.
- Enter the Z-axis feed rate.
- If you want the tool to retract:

Enter either the number of retracts OR the distance between each retract.

Enter the length of time (in seconds) the quill should dwell (pause) during the retract cycle.

Enter the lenght of time (in seconds) the tool should dwell at the end depth before the final retract.

- Press the **USE** key.

Drill - The quill will feed down then rapidly move up.

Bore - The quill will feed down then feed up.

Position - You will be prompted to move the quill manually.

Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running your program (refer to **Programming a Tool Step**).

Center Line

With the position/drill feature, you can also locate the midpoint of two points or the center line of a circle.

To locate a center line:

- Press the **CENTER LINE** softkey.
- Locate the first edge then press the **ENTER** key.
- Locate the second edge of your part then press the **ENTER** key.
- If you're calculating the center of a circle, locate then enter three points along the diameter of the circle.
- Press the **USE** key.

If you use an electronic edge finder, the points will automatically be entered on contact—even if you over-travel.

Line

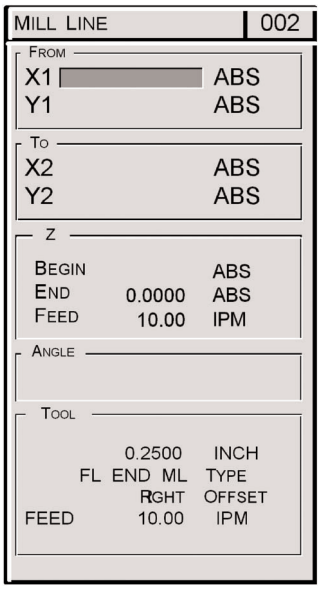
Lines are defined by their “From” point (the point where they begin) and “To” point (the point where they end).

There are two ways you can program a line:

- With four coordinates (X1, Y1, X2, Y2)
- With three of the coordinates above (X1, X2, Y2 or X1, Y1, X2, etc.) and an angle

Choose a method based upon the information available from your print.

To program a line:

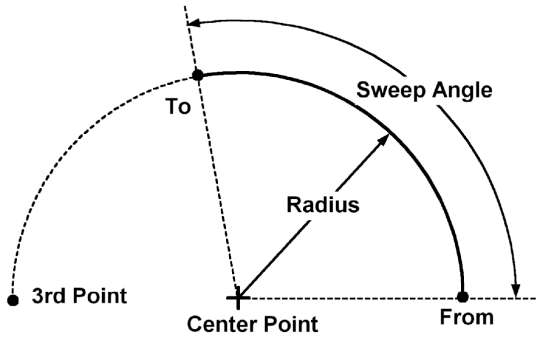


- Press the **LINE** key.
- Enter the beginning X- and/or Y-axes coordinates into the “From” field.
- Enter the ending X- and/or Y-axes coordinates into the “To” field.
- Enter the begin and end depths for the Z-axis.
- Enter the Z-axis feed rate.
- If one of the X- or Y-axes fields above was left blank, enter an angle.
- Highlight the "Offset" field and press the **LEFT**, **CENTER,** or **RIGHT** softkey.
- Enter the table’s feed rate.
- Press the **USE** key.

Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running your program (refer to **Programming a Tool Step**).

Arc

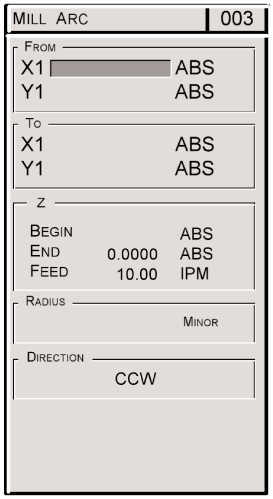
An arc can be defined several ways:



- With a From point, To point and a radius
- With a From point, To point and a center point
- With a From, To and a 3rd point along the arc
- With a From or To point, center point and a sweep angle

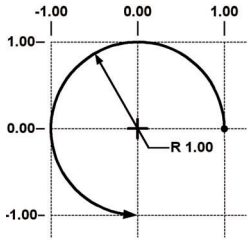
Choose a method based upon the information available from your print. While programming, keep in mind that the arc's sweep angle is measured from the X-axis.

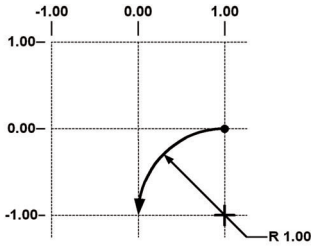
To program an arc:



- Press the **ARC** key.
- Enter the beginning coordinates for the X-axis (X1) and Y-axis (Y1) in the “From” field.
- Enter the ending coordinates for the X-axis (X2) and Y-axis (Y2) in the “To” field.
- Enter the begin and end depths for the Z-axis.
- Enter the Z-axis feed rate.
- Enter the arc's radius, then press either the **MAJOR ARC** or **MINOR ARC** softkey. (A major arc has a sweep angle greater than 180 degrees; a minor arc’s sweep angle is less than 180 degrees.)

MAJOR ARC





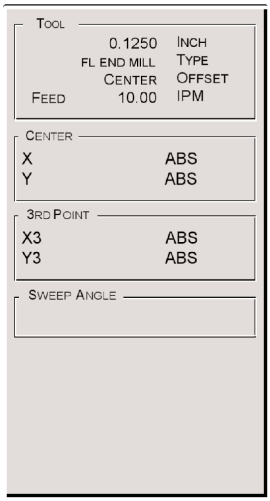
MINOR ARC

- Select the cutting direction. Press the **CW** softkey for a clockwise direction or the **CCW** softkey for a counter-clockwise direction.

- Arrow down and highlight the "Offset" field. Using the softkeys, select the tool offset—**LEFT**, **CENTER**, **RIGHT**, **INSIDE** or **OUTSIDE**.

- Enter the table's feed rate.

- If you need to enter a center coordinate, 3RD point and/or sweep angle:



  Center
  Enter the center coordinate's position for the X- and Y-axes.

  3RD Point
  Enter your 3RD coordinate's position for the X-axis (X3) and Y-axis (Y3).

  Sweep Angle
  Enter the sweep angle.

  Information that appears in blue has been calculated. If any of these values are already displayed in blue, then **MILL*PWR*** has enough data for the arc and has calculated the rest.

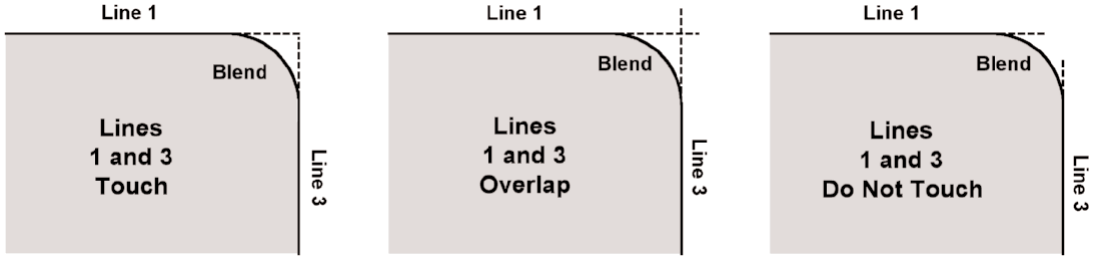
- Press the **USE** key.

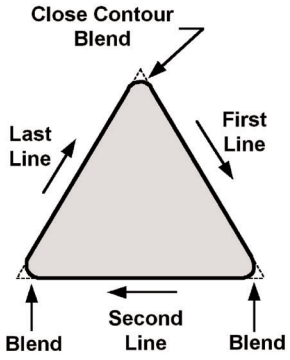
Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running your program (refer to **Programming a Tool Step**).

Blend

A blend is an arc that connects two lines, two arcs or a line and an arc. All you have to do is provide the radius for the blend and indicate whether it is normal or inverted. **MILL*PWR*** will calculate the tangent points for you.

The two steps you want to blend can, but don't have to, intersect or touch. If they don't come into contact with each other, check that your radius is large enough to connect them.





It's also possible to close a contour (e.g., a triangle) using the blend feature by inserting a blend step immediately after the last step in the contour.

After you enter the blend's radius, press the **CLOSE CONTOUR** softkey, and **MILL*PWR*** will blend the last step with the first step.

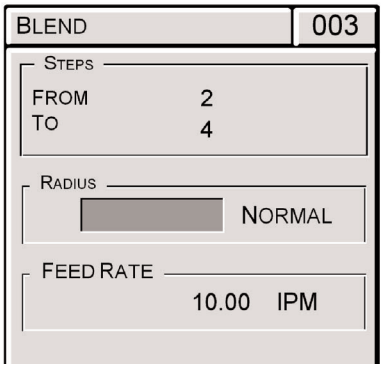
To program a blend:

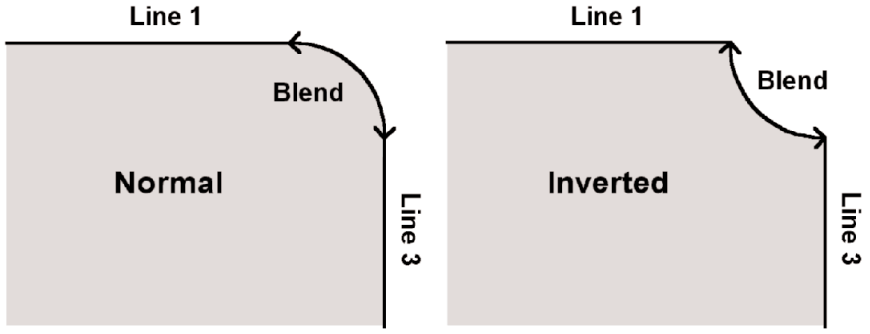
- Highlight a step within your program where you want to place a blend.
- Press the **BLEND** key.

- Check that the steps listed in the “From” and “To” fields are the steps you want to blend. (If they're incorrect, press the **CANCEL** key and highlight the appropriate step.)

- Enter the blend's radius. (Press the **CLOSE CONTOUR** softkey if you want to blend the end of a contour with the beginning. The step numbers in the "To" and "From" fields will automatically change.)

- Press either the **NORMAL ARC** or **INVERTED ARC** softkey. A normal arc curves outward; an inverted arc curves inward.

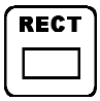




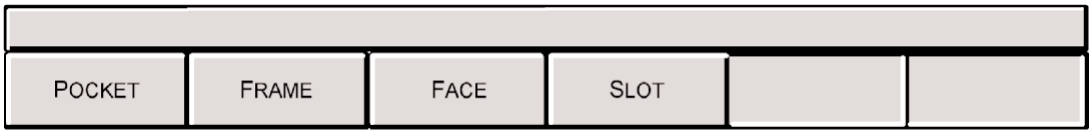
- Enter the table's feed rate.

- Press the **USE** key.

Rectangular Milling Functions

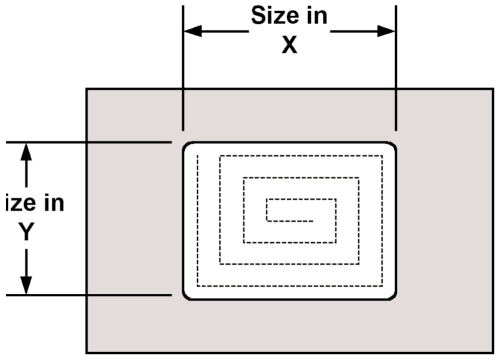


MILL*PWR* offers several rectangular milling functions that let you program pockets, frames, faces and slots quickly and easily.



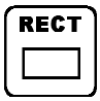
Pocket

A pocket is a cavity or area on your part where material is removed when you machine. You can program a rectangular pocket two ways:

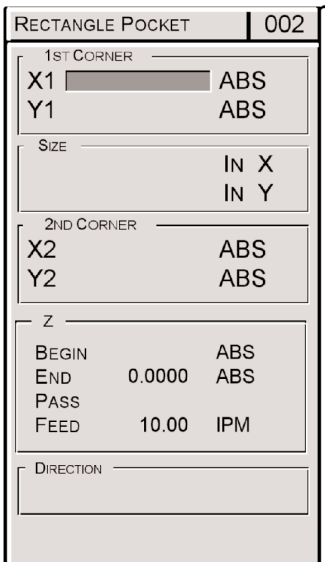


- Using the coordinates of two diagonal corners.
- Using the coordinates of one corner and the size of the pocket.

<u>To program a rectangular pocket:</u>



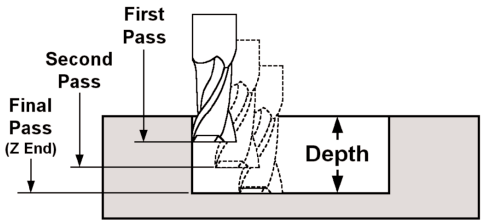
- Press the **RECT** key.
- Press the **POCKET** softkey.
- Enter the X- and Y-axes coordinates for the pocket's 1ST Corner.
- Now enter either the size of the pocket or the coordinates for the 2ND Corner.



Size
Enter the length of the pocket along the X- and Y-axes.

2ND Corner
Enter the X- and Y-axes coordinates for the 2ND Corner. (The 2ND Corner must be located diagonally from the 1ST Corner.)

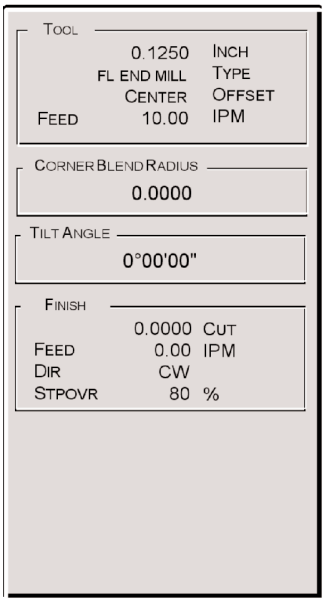
- Enter the begin and end depths for the Z-axis.



- Enter either the number of passes OR the distance between each pass. “Pass” refers to the cuts that are used to machine the pocket to its “End” depth.
- Enter the Z-axis feed rate.
- For “Direction,” press either the **CW** softkey for a clockwise cutting direction or the **CCW** softkey for a counter-clockwise cutting direction.
- Arrow down and enter the table’s feed rate.

The corner blend radius defaults to the last radius entered. Check the value to ensure that it is correct.

- If you need to program a corner blend radius, tilt angle and/or finish cut:



Corner Blend Radius

You can add a corner blend radius to the corners of a rectangular pocket.

Highlight the “Corner Blend Radius” field and enter a radius.

Tilt Angle

You can tilt a rectangular pocket by identifying a tilt angle.

Highlight the “Tilt Angle” field and enter an angle (measured from the X-axis).

Finish

Finish allows you to leave some excess material that will be removed during the finish cut reducing, if not eliminating tool marks. The finish cut will automatically arc on and arc off.

Enter the amount of material to be removed during the finish cut.

Enter the feed rate for the finish cut.

Select the finish cut's direction. Press the **CW** softkey for a clockwise direction or the **CCW** softkey for a counter-clockwise direction.

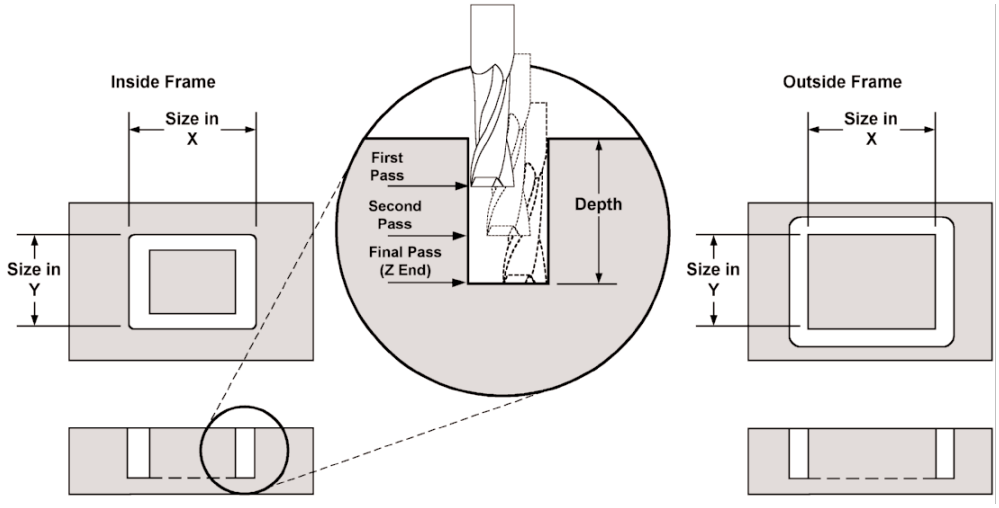
Enter a stepover percentage (how much you want your tool to overlap on each pass).

- Press the **USE** key.

Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running your program (refer to **Programming a Tool Step**).

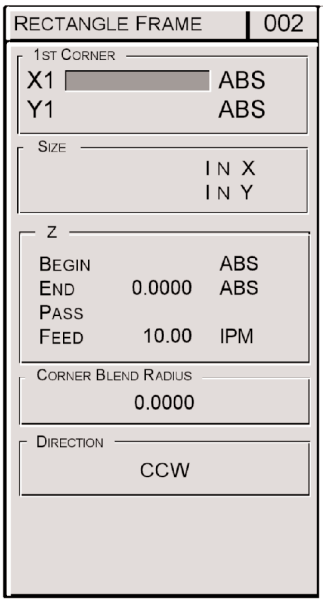
Frame

When you program a rectangular frame, you define it by its first corner, and its size or diagonal corner.



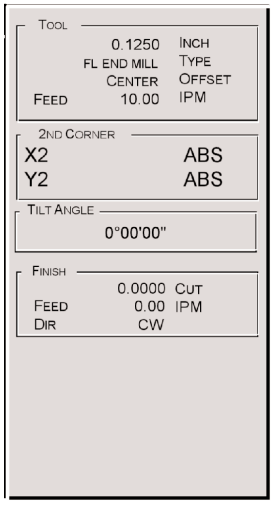
To program a rectangular frame:

- Press the **RECT** key.
- Press the **FRAME** softkey.
- Enter the X- and Y-axes coordinates for the frame's 1ST Corner.
- In the "Size" field, enter the length of the frame along the X- and Y-axes (unless you're programming a 2ND Corner).
- Enter the begin and end depths for the Z-axis.
- Enter either the number of passes OR the distance between each pass. "Pass" refers to the cuts that are used to machine the pocket to its "End" depth.
- Enter the Z-axis feed rate.
- (Optional) Enter a corner blend radius.



- For “Direction,” press either the **CW** softkey for clockwise or the **CCW** softkey for a counter-clockwise direction.

- Arrow down and select a tool offset by pressing the appropriate softkey.



- Enter the table’s feed rate.

- If you need to program a 2ND Corner, tilt angle and/or finish cut:

  2ND Corner
  Enter the X- and Y-axes coordinates for the 2ND Corner. (The 2ND Corner must be located diagonally across from the 1ST Corner.)

  Tilt Angle
  Enter the tilt angle.

  Finish
  Highlight "Cut" in the“Finish” field. Enter the amount of material to be removed during the finish cut.

  Enter the feed rate for the finish cut.

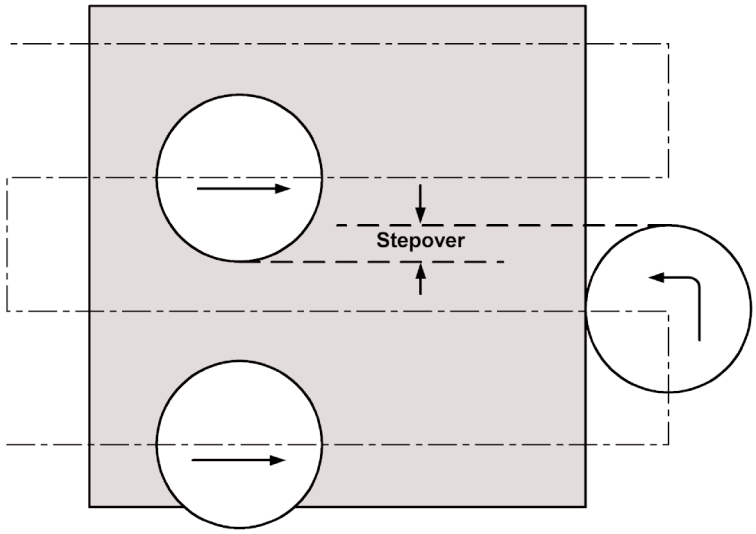
  For “Direction,” press either the **CW** softkey for clockwise or the **CCW** softkey for a counter-clockwise direction.

- Press the **USE** key.

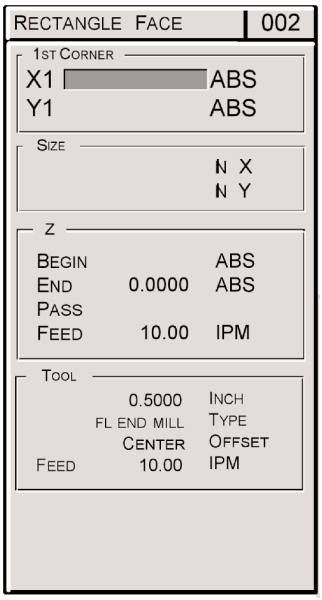
Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running your program (refer to **Programming a Tool Step**).

Face

The “Rectangle Face” step provides a quick way to face off your workpiece. Simply enter the coordinates from one corner and either the size of the area to be faced off or the coordinates for a diagonal corner. **MILL*PWR*** will position your table at the lower left end of the area you've programmed.

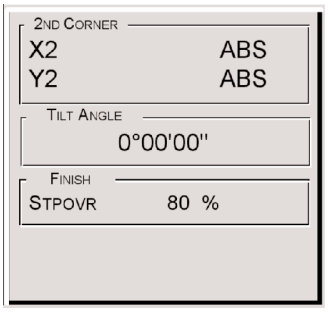


To program a rectangle face:



- Press the **RECT** key.
- Press the **FACE** softkey.
- Enter the X- and Y-axes coordinates for the face's 1ST Corner.
- In the “Size” field, enter the length of the face along the X- and Y-axes (unless you’re programming a 2ND Corner).
- Enter the begin and end depths for the Z-axis.
- Enter either the number of passes OR the distance between each pass. “Pass” refers to the cuts that are used to machine the pocket to its “End” depth.

- Enter the Z-axis feed rate.
- Select a tool offset by pressing the appropriate softkey.
- Enter the table's feed rate.
- If you wish to program a 2ND Corner and/or a tilt angle or would like to program a finish cut, press the **MORE** softkey.



2ND Corner
Enter the X- and Y-axes coordinates for the 2ND Corner. (The 2ND Corner must be located diagonally across from the 1ST Corner.)

Tilt Angle
Enter the tilt angle.

Finish
Enter a stepover percentage (how much you want your tool to overlap on each pass).

- Press the **USE** key.

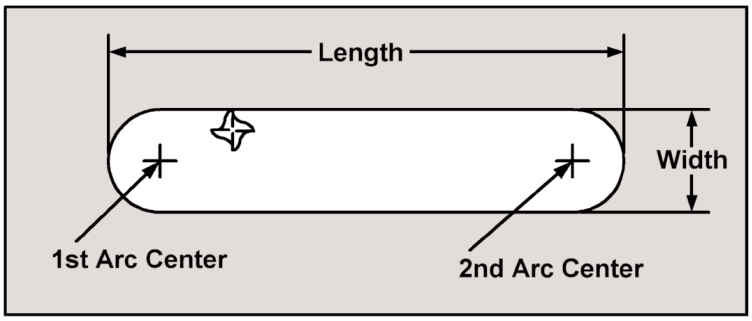
Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running your program (refer to **Programming a Tool Step**).

Slot

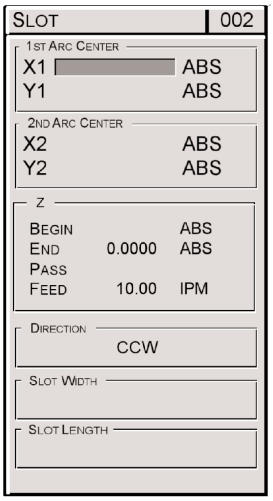
You can program a slot two ways:

- By entering the center point of each arc and the slot's width
- By entering the center point of one arc, the length and width of the slot, and an angle

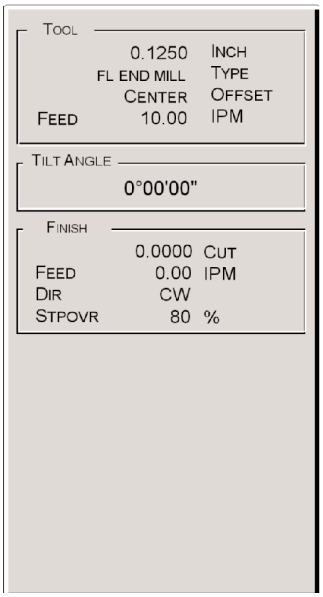
Choose a method based upon the information available from your print.



To program a slot:



- Press the **RECT** key.
- Press the **SLOT** softkey.
- Enter the X- and Y-axes coordinates for the "1ST Arc Center."
- Enter the X- and Y-axes coordinates for the "2ND Arc Center" or arrow down and enter the "Slot Length" and "Angle."
- Enter the begin and end depths for the Z-axis.
- Enter either the number of passes OR the distance between each pass. "Pass" refers to the cuts that are used to machine the pocket to its "End" depth.
- Enter the Z-axis feed rate.
- Select the direction you want your arc to be cut. Press the **CW** softkey for clockwise or the **CCW** softkey for a counter-clockwise direction.



- Enter the slot's width.
- Enter the slot's length.
- Arrow down and enter the feed rate.
- If you want to program a tilt angle and/or finish cut:

  Tilt Angle

  You can slant a slot by identifying a tilt angle.

  Highlight the "Tilt Angle" field and enter an angle (measured from the X-axis).

  Finish

  Enter the amount of material to be during the finish cut.

  Enter the feed rate for the finish cut.

  Select the finish cut's direction. Press the **CW** softkey for a clockwise direction or the **CCW** softkey for a counter-clockwise direction.

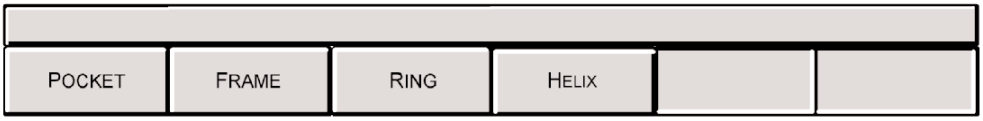
  Enter a stepover percentage (how much you want your tool to overlap on each pass).

- Press the **USE** key.

Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running your program (refer to **Programming a Tool Step**).

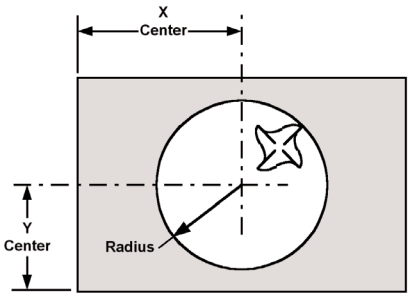
Circular Milling Functions

MILL*PWR* offers several circular milling functions that let you program pockets, frames and rings quickly and easily.



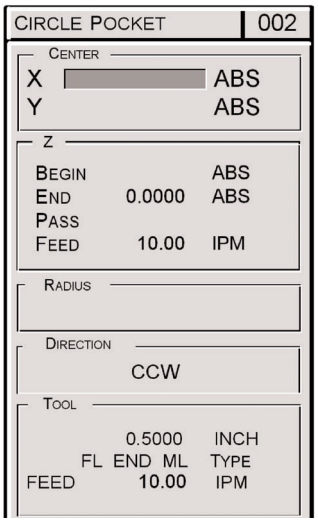
Pocket

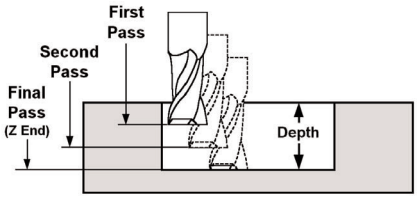
A pocket is a cavity or area on your part where material is removed when you machine. You can program a circular pocket by indicating the center point and radius.



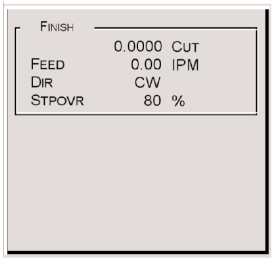
To program a circular pocket:

- Press the **CIRCLE** key.
- Press the **POCKET** softkey.
- Enter the X- and Y-axes coordinates for the pocket's center point.
- Enter the begin and end depths for the Z-axis.
- Enter either the number of passes OR the distance between each pass. “Pass” refers to the cuts that are used to machine the pocket to its “End” depth.
- Enter the Z-axis feed rate.
- Enter the radius.
- For the "Direction,” press the **CW** softkey for a clockwise direction or the **CCW** softkey for a counter-clockwise direction.
- Enter the cutting feed rate.





- If you want to program a finish cut, press the **MORE** softkey.



  Enter the amount of material to be removed during the finish cut.

  Enter the feed rate for the finish cut.

  Select the finish cut's direction. Press the **CW** softkey for a clockwise direction or the **CCW** softkey for a counter-clockwise direction.

  Enter a stepover percentage (how much you want your tool to overlap on each pass).

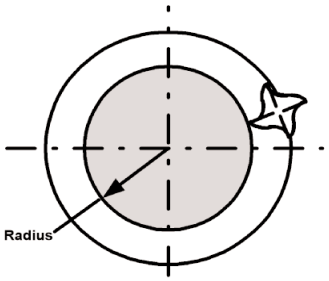
- Press the **USE** key.

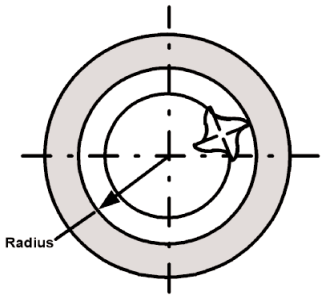
Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running your program (refer to **Programming a Tool Step**).

Frame

A circular frame is determined by its center point and radius. The direction will determine if you're climb cutting or conventional cutting. The tool offset will determine if you are cutting an inside or outside frame.

OUTSIDE FRAME

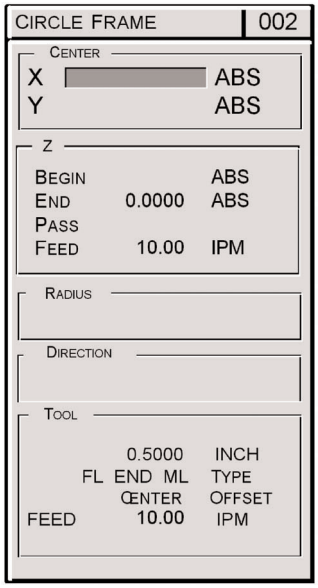


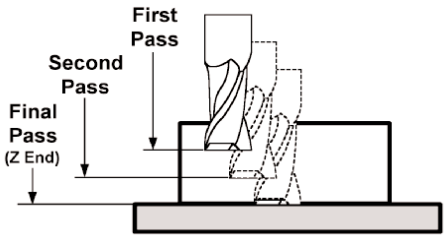


INSIDE FRAME

To program a circular frame:

- Press the **CIRCLE** key.
- Press the **FRAME** softkey.
- Enter the X- and Y-axes coordinates for the center point.
- Enter the begin and end depths for the Z-axis.
- Enter either the number of passes OR the distance between each pass. "Pass" refers to the cuts that are used to machine the pocket to its "End" depth.



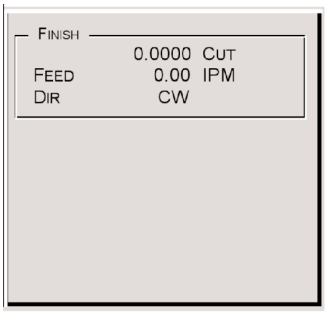


- Enter the Z-axis feed rate.
- Enter the radius.
- For the direction, press the **CW** softkey for clockwise or the **CCW** softkey for a counter-clockwise direction.
- Select the appropriate tool offset.
- Enter the table's feed rate.
- If you would like to program a finish cut, press the **MORE** softkey.

  Enter the amount of material to be removed during the finish cut.

  Enter the feed rate for the finish cut.

  Select the finish cut's direction. Press the **CW** softkey for a clockwise direction or the **CCW** softkey for a counter-clockwise direction.



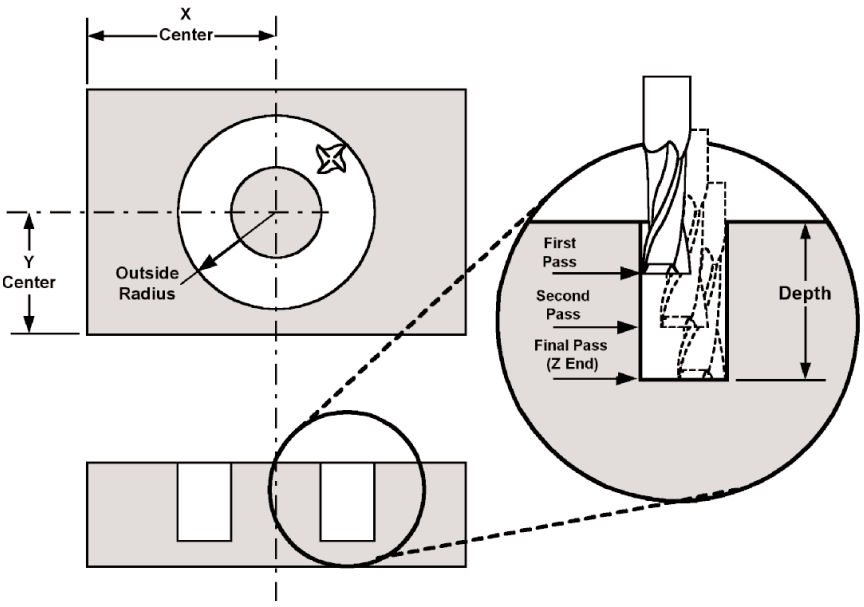
- Press the **USE** key.

Note: If the tool size and type in the "Tool" field are incorrect, change the tool settings before running your program (refer to **Programming a Tool Step**).

Ring

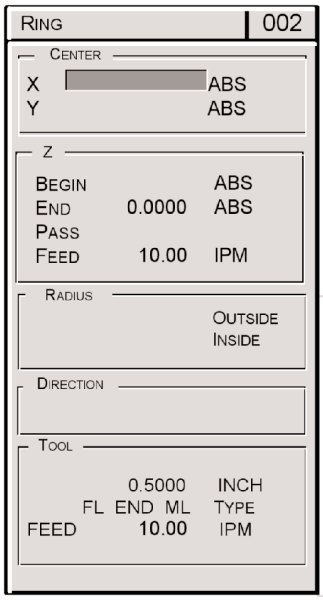
A ring is a circular pocket with a circular island in the center. A ring is determined by its center point, outside radius (radius of the pocket) and inside radius (radius of the island).

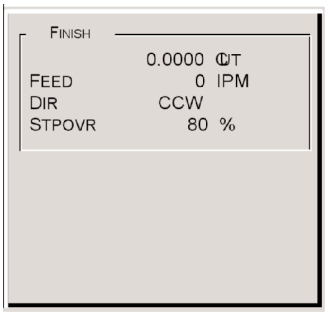
The direction of the cut on the inside radius will determine whether you are climb cutting or conventional cutting. **MILL*PWR*** will reverse the tool direction on the outside radius so that the cutting direction stays the same.



To program a ring:

- Press the **CIRCLE** key.
- Press the **RING** softkey.
- Enter the X- and Y-axes coordinates for the center point.
- Enter the begin and end depths for the Z-axis.
- Enter either the number of passes OR the distance between each pass. “Pass” refers to the cuts that are used to machine the pocket to its “End” depth.
- Enter the Z-axis feed rate.
- Enter the circle's outside and inside radii.





- For “Direction,” press the **CW** softkey for clockwise or the **CCW** softkey for a counter-clockwise direction.
- Enter the table’s feed rate.
- If you would like to program a finish cut, press the **MORE** softkey.

  Enter the amount of material to be removed during the finish cut.

  Enter the feed rate for the finish cut.

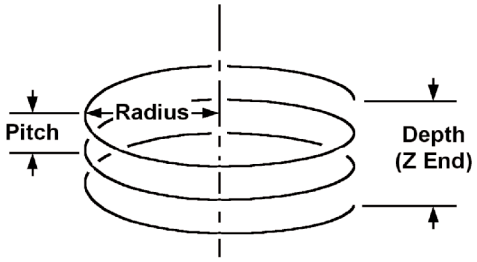
  Select the finish cut's direction. Press the **CW** softkey for clockwise or the **CCW** softkey for a counter-clockwise direction.

  Enter a stepover percentage (how much you want your tool to overlap on each pass).
- Press the **USE** key.

Note: If your tool size and type listed in the "Tool" field are incorrect, change the tool settings before running your program (refer to **Programming a Tool Step**).

Helix

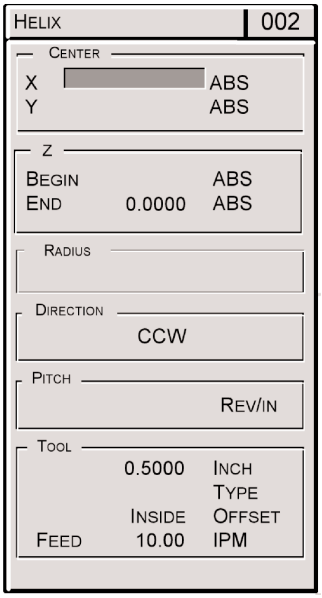
A helix is a spiral-like form that winds at a uniform angle. A helix can be defined two ways:



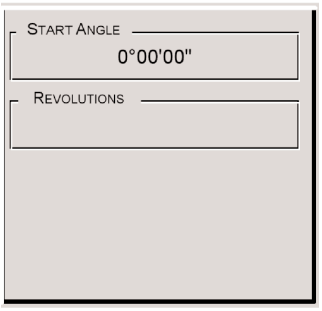
- By the radius, depth and pitch
- By the radius, depth and number of revolutions

To program a helix:

- Press the **CIRCLE** key.
- Press the **HELIX** softkey.
- Enter the X- and Y-axes coordinates for the center point.
- Enter the begin and end depths for the Z-axis.
- Enter the radius.
- For "Direction," press the **CW** softkey for clockwise or the **CCW** softkey for a counter-clockwise direction.
- (Optional) Enter the pitch.
- Choose **INSIDE** or **OUTSIDE** offset.
- Enter the table's feed rate.



- If you need to program a start angle and/or a number of revolutions, press the **MORE** softkey.



  Start Angle
  Enter the angle where the helix begins (3 o'clock position is 0 degrees; 12 o'clock position is 90 degrees).

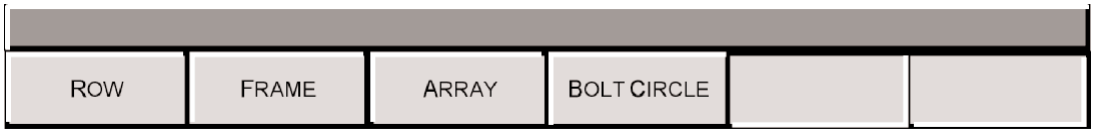
  Revolutions
  Enter the number of revolutions.

- Press the **USE** key.

Note: If your tool size and type listed in the "Tool" field are incorrect, change the tool settings before running your program (refer to **Programming a Tool Step**).

Hole Patterns

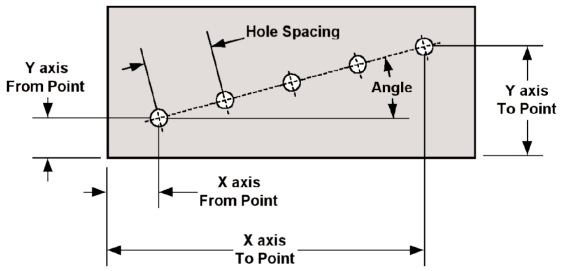
MILL*PWR* includes several built-in routines that let you program hole patterns quickly and easily.



Row of Holes

A row of holes, can be programmed two ways:

- By entering the coordinates of the first and last hole
- By entering the coordinates of the first hole, the spacing between each hole and the row's angle



The "From" point refers to the center of the first hole, while the "To" point is the center of the last hole. Any additional holes will be spaced equally between these two.

You'll also be prompted for peck and tool retract values. "Peck" lets you break chips and reduce chip buildup during drilling operations. "Tool Retract" allows you to program **MILL*PWR*** to raise the tool at regular intervals.

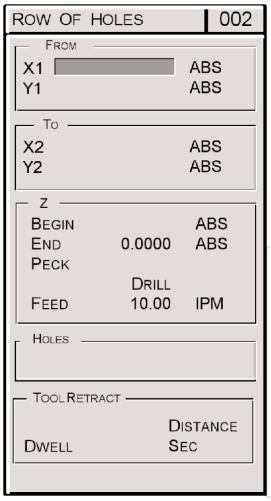
To program a row of holes:

- Press the **HOLES** key.
- Press the **ROW** softkey.
- In the "From" field, enter the X- and Y-axes coordinates for the center of the first hole.

- Now either:

  Enter the X- and Y-axes coordinates for the center of the last hole in the "To" field; or

  Press the **MORE** softkey and enter the distance you want between each hole (from center point to center point) in the "Hole Spacing" field. Also enter the angle of the row of holes.



- Enter the begin and end depths for the Z-axis.

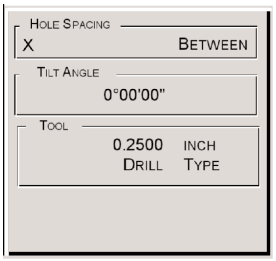
- (Optional) Enter either the number of pecks OR the distance between each peck.

Drill - The quill will feed down then rapidly move up.

Bore - The quill will feed down then feed up.

Position - You will be prompted to move the quill manually.

- Select the **DRILL**, **BORE** or **POSITION** softkey.



- Enter the Z-axis feed rate.

- Enter the number of holes you want to include in the row.

- If you want the tool to retract:

  Enter either the number of retracts OR the distance between each retract.

  Enter the length of time (in seconds) that you want the quill to dwell (pause) during each retract.

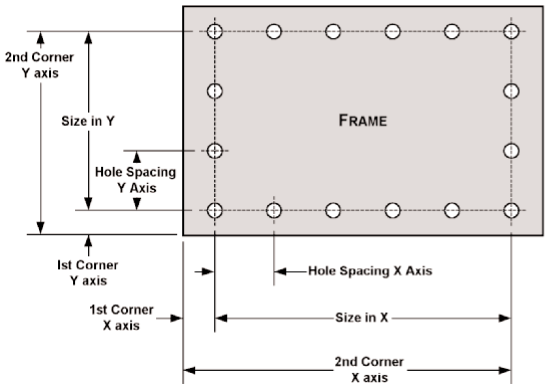
- Press the **USE** key.

Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running your program (refer to **Programming a Tool Step**).

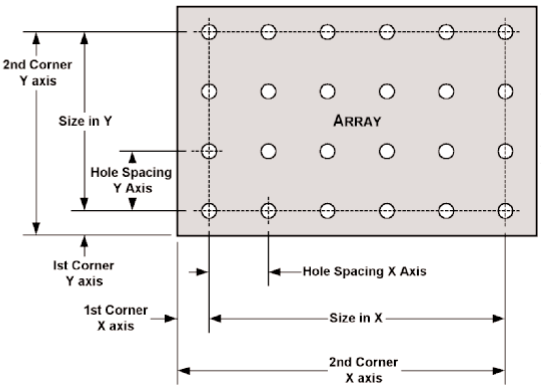
Hole Frame and Hole Array

Hole frame and hole array patterns require the same information, but their patterns differ slightly. Hole frames limit holes to the outside edge of a rectangular shape, while hole arrays allow holes along the outside edge and throughout the center.

Hole Frame



Hole Array

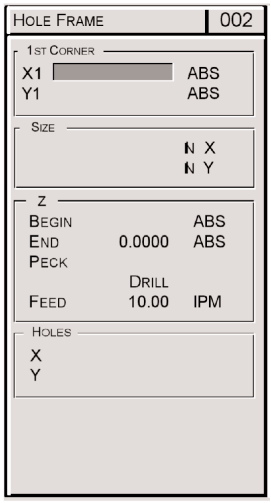


Hole frames and hole arrays can be defined three ways:

- By the position of the 1ST Corner, size, and the number of holes
- By the position of the 1ST Corner, position of the 2ND (diagonal) Corner, and number of holes
- By the position of the 1ST Corner, hole spacing, and number of holes

Choose the method that's easiest for you based upon the information from your print.

To create a hole frame or hole array:



- Press the **HOLES** key.
- Press the **FRAME** softkey to program a hole frame; press the **ARRAY** softkey to program a hole array.
- Enter the X- and Y-axes coordinates for your 1ST corner into the "From" field.
- In the “Size” field, enter the lengths along the X- and Y-axes.
- Enter the begin and end depths for the Z-axis.
- (Optional) Enter either the number of pecks OR the distance between each peck.
- Select the **DRILL**, **BORE** or **POSITION** softkey.
- Enter the Z-axis feed rate.
- In the "Holes" field, enter the number of holes you want to include along the X- and Y-axes.
- If you would like to program a retract, 2ND corner, hole spacing and/or tilt angle, press the **MORE** softkey.

Drill - The quill will feed down then rapidly move up.

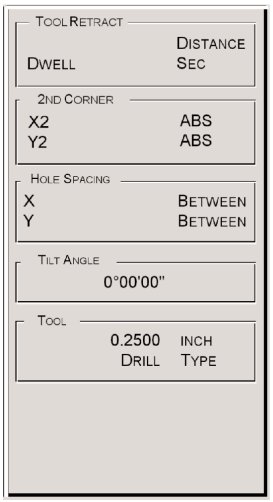
Bore - The quill will feed down then feed up.

Position - You will be prompted to move the quill manually.

Tool Retract

Enter either the number of retracts OR the distance between each retract.

Enter the length of time (in seconds) that you want the quill to dwell (pause) during each retract.



2ND Corner
Enter the X- and Y-axes coordinates for the 2ND Corner (located diagonally from the 1ST corner).

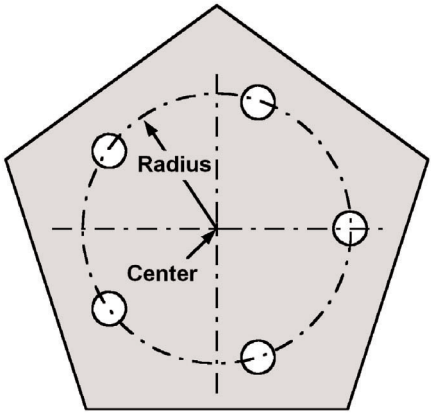
Hole Spacing
Specify the spacing between holes along the X- and Y-axes.

Tilt Angle
Enter a tilt angle, if needed.

- Press the **USE** key.

Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running your program (refer to **Programming a Tool Step**).

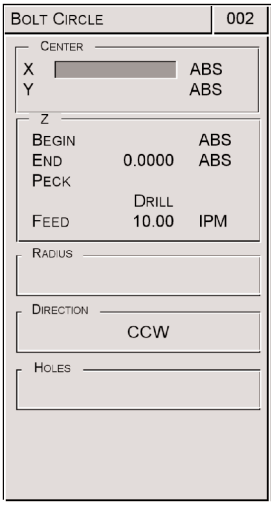
Bolthole Circle Patterns



A bolthole circle pattern is defined by its center point, radius and number of holes. You can program partial bolthole patterns by pressing the **MORE** softkey and entering a start angle and an end angle.

To program a bolthole circle pattern:

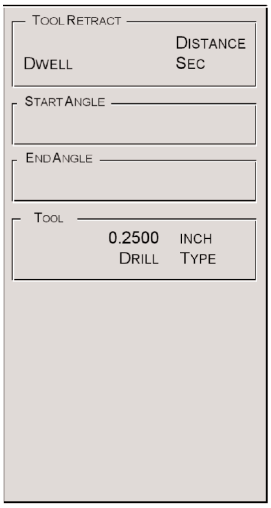
- Press the **HOLES** key.
- Press the **BOLT CIRCLE** softkey.
- Enter the X- and Y-axes coordinates for the bolthole circle pattern's center point.
- Enter the begin and end depths of the cut for the Z-axis.
- (Optional) Enter either the number of pecks OR the distance between each peck.
- Select the **DRILL**, **BORE** or **POSITION** softkey.
- Enter the Z-axis feed rate.
- Enter the radius of the hole pattern.
- For "Direction," select the direction you want to cut. Press the **CW** softkey for clockwise or the **CCW** softkey for a counter-clockwise direction.
- Enter the number of holes you want to include in the circle.



Drill - The quill will feed down then rapidly move up.

Bore - The quill will feed down then feed up.

Position - You will be prompted to move the quill manually.



- If you need to program a tool retract, start angle and/or end angle, press the **MORE** softkey.

  Tool Retract

  Enter the number of tool retracts or the distance between each retract. Enter the length of time (in seconds) that you want the tool to dwell (pause).

  Start Angle

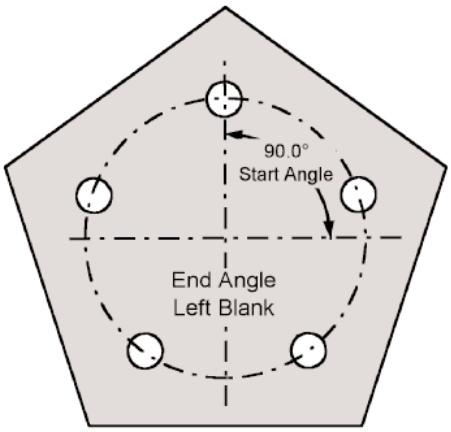
  Enter the start angle of the first hole in the bolthole circle pattern (refer to graphic, below).

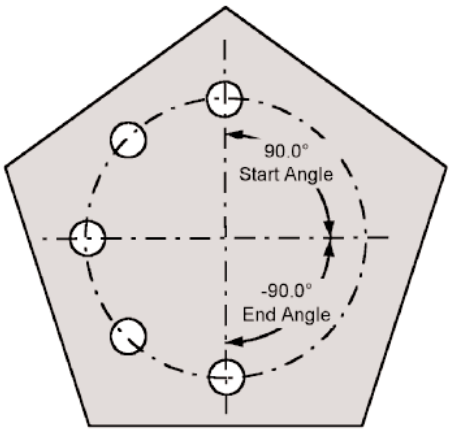
  End Angle

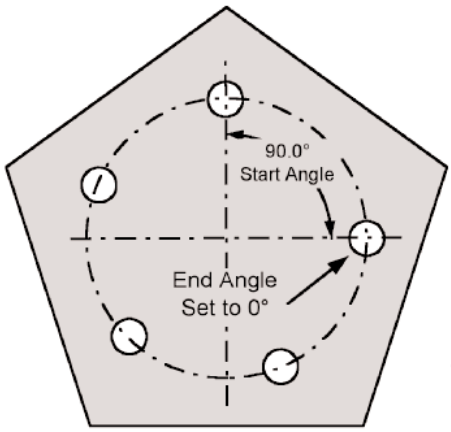
  Enter the end angle of the last hole in the bolthole circle pattern (refer to graphic, below).

- Press the **USE** key.

Note: If the tool size and type in the "Tool" field are incorrect, change the tool settings before running your program (refer to **Programming a Tool Step**).

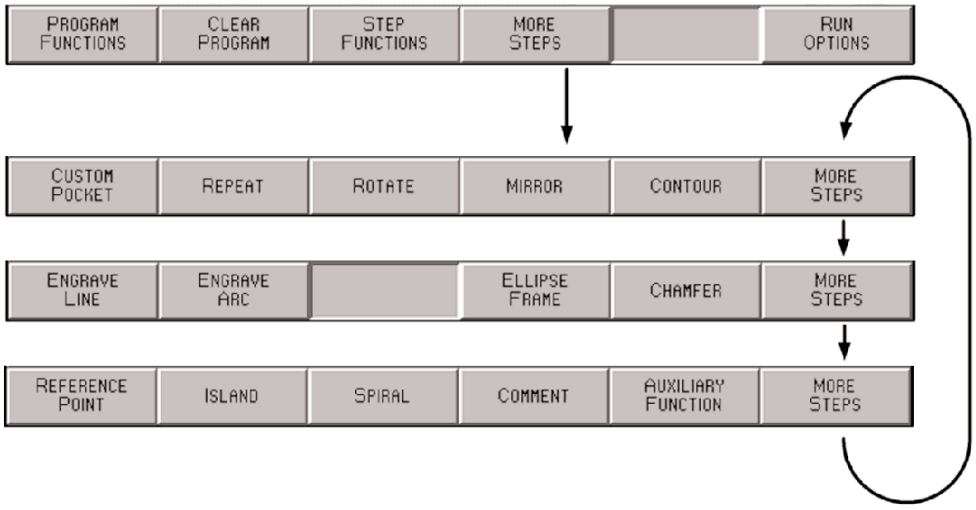






Additional Milling Functions

Additional milling functions are available from the PGM screen by pressing the **MORE STEPS** softkey.



Custom Pocket

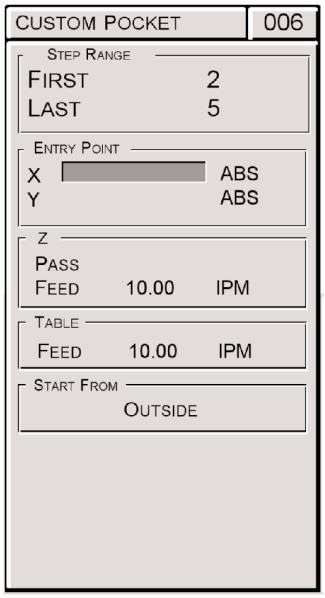
Condition: The custom pocket step must immediately follow a closed contour.

You can create a custom pocket from any closed contour. A closed contour is any shape consisting of lines, arcs, and/or blends, where the last step ends at the same point where the first step begins. **MILL*PWR*** will indicate a closed contour with double lines to the right of the applicable steps in the program list.

The "Custom Pocket" step must be placed immediately following the last step of the closed contour. **MILL*PWR*** will automatically fill in the step range for you. You'll still need to fill in the "Entry Point," which is the plunge point for the tool, and set the feed rate for the custom pocket. Custom pockets can be cut from the center (inside) of the pocket out or from the outside edge of the pocket in (towards the center). The finish cut will occur after the material from the inside of the pocket has been removed.

The cut direction is determined by the order with which you programmed the individual contour steps.

To program a custom pocket:



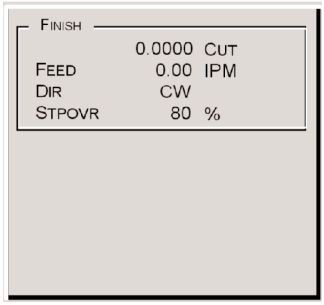
- Create a closed tool path.
- Position the cursor immediately below the closed tool path.
- Press the **MORE STEPS** softkey.
- Press the **CUSTOM POCKET** softkey. **MILL*PWR*** will automatically fill in the step range for you.
- Enter the X- and Y-axes coordinates for the entry (plunge) point.
- Enter either the number of passes OR the distance between each pass. “Pass” refers to the cuts that are used to machine the pocket to its “End” depth.
- Enter the feed rate for the Z-axis.
- Enter the table’s feed rate.
- Press either the **OUTSIDE** or **INSIDE** softkey. “Outside” clears the custom pocket from the outside in. “Inside” clears the custom pocket from the inside out.
- Press **USE** or continue on to finish cut.
- If you would like to program a finish cut, press the **MORE** softkey, then:

  Enter the amount of material to be removed during the finish cut in the “Cut” field.

  Enter the feed rate for the finish cut.

  For “Direction,” press either the **CW** softkey for clockwise or the **CCW** softkey for a counter-clockwise direction.

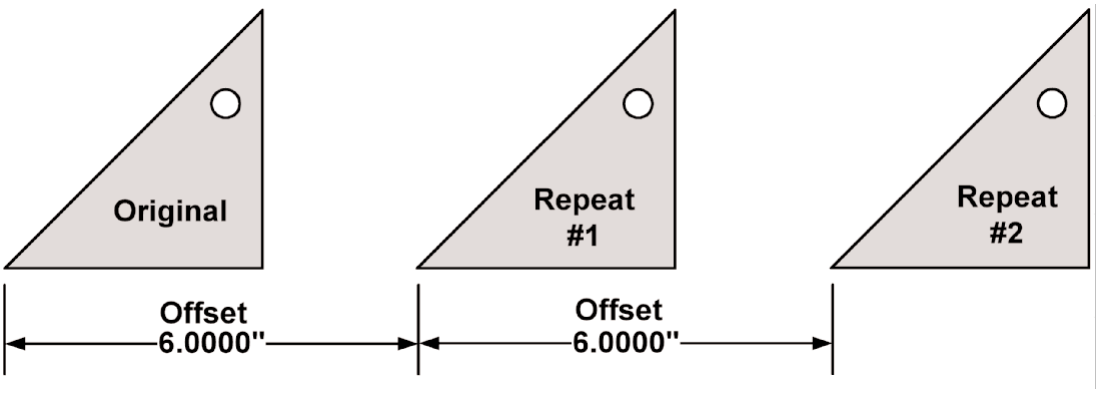
  Enter the stepover percentage (how much you want your tool to overlap on each pass).



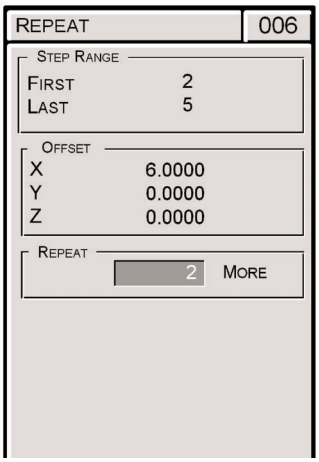
- Press the **USE** key.

Repeat

Using this step you can repeat whole programs or sections of programs horizontally, vertically or both.



To program a repeat:



- From the PGM screen, press the **MORE STEPS** softkey.
- Press the **REPEAT** softkey.

Tip: Use the **TEACH POSITION** softkey to indicate a second.

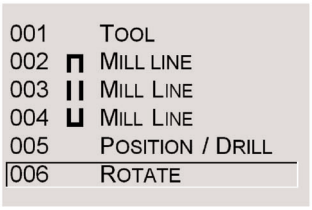
- Enter the number of the first step and the last step in the step range that you want to repeat.
- Enter the offset for the X-, Y- and/or Z-axis. (The *offset* is the distance between repeats.)
- Enter the number of times you want to repeat the original steps.

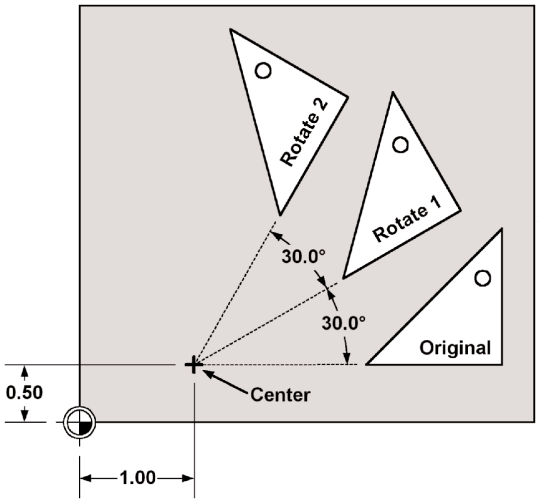
Note: The steps being repeated must precede the “Repeat” step.

Tip: Entering a 0(zero) for the number of repeats will change the location of the original steps entered in the FROM and TO fields. To shift the datum of an entire program, place a repeat step at the end of the program and enter the required coordinate shift(s) in the X-, Y-, and/or Z-axis fields. When USE is pressed the entireprogram will shift by the entered values.

Rotate

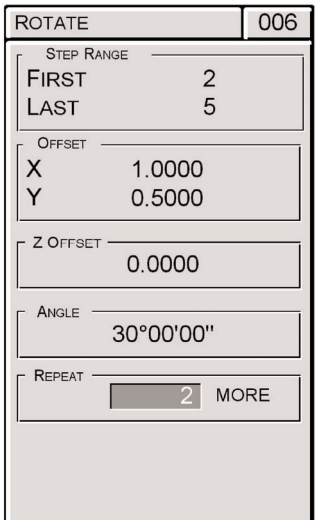
With “Rotate,” you can rotate whole programs or sections of programs.





Condition: The steps rotated must precede the “Rotate” step.

To program a rotate:

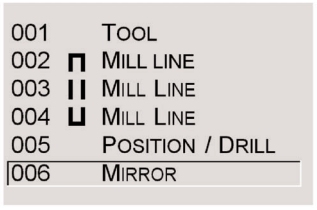


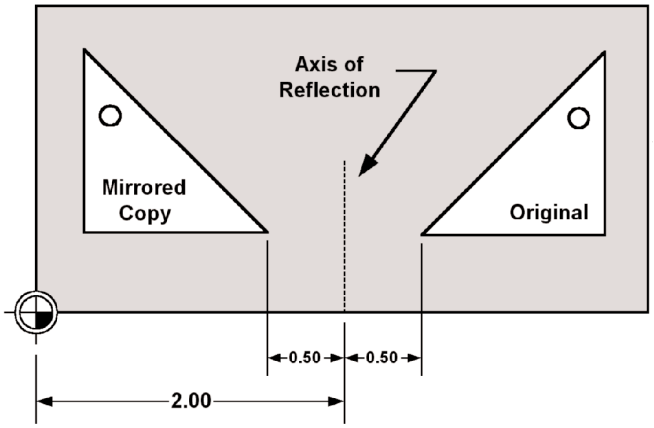
- From the PGM screen, press the **MORE STEPS** softkey
- Press the **ROTATE** softkey.
- Enter the first and last steps in the range of steps that you would like to rotate.
- Enter the X- and Y-axes coordinates for the center point of rotation.
- Enter an offset for the Z-axis.
- Enter the angle for each rotation.
- Enter the number of times you want to rotate the shape. If you just want to rotate the original program or sections of your program you’re working with, enter “0.” If you want additional rotations of your original, enter the number of rotations that you want in addition to your original.
- Press the **USE** key.

Mirror

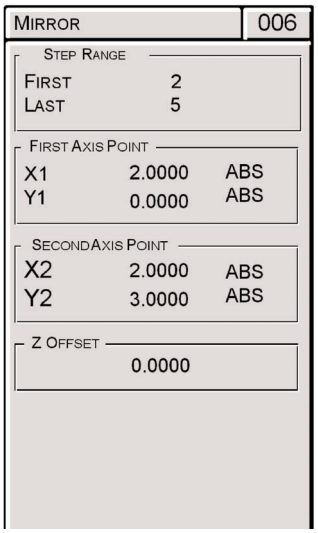
Condition: The steps being mirrored must precede the "Mirror" step.

With "Mirror," you can create a mirror image of an entire program or a section of a program.





To program a mirror:



- From the PGM screen, press the **MORE STEPS** softkey.
- Press the **MIRROR** softkey.
- Enter the number of the first step and the last step in the step range that you want to mirror.
- Define the axis of reflection (a line that separates the mirrored image from the original one).

  For "1ST Axis Point," enter the X- and Y-axes coordinates for the axis of reflection.

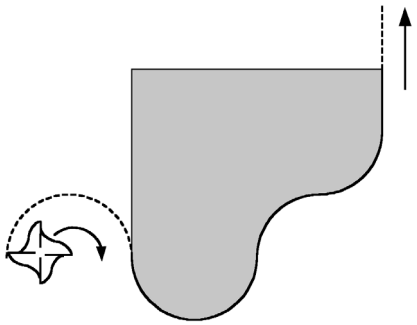
  For "2ND Axis Point," enter the X- and Y-axes coordinates for the axis of reflection.
- Enter an offset for the Z-axis.
- Press the **USE** key.

Contour

The "Contour" step enables you to approach and/or depart from your part on a straight line or with an arc.

Conditions: 1) The contour step must immediately follow the contour steps.
2) Contours can only be associated with lines and arcs.

By adding contours before and/or after a continuous tool path, you'll avoid starts and stops striking against the workpiece edge.



With an arc approach/departure, the tool will take a rounded turn as it nears or exits the workpiece. (The curved dotted line on the left of the graphic represents an arcing approach to the workpiece.)

With a straight approach/departure, the tool path is extended away from the workpiece. (The dotted line on the right of the graphic shows a straight contour at the end of the cutting path.)

The step range can include one or more steps. If you're planning to add a contour to an individual step, the first and last steps in the range will be the same.

Because the approach and departure fields are independent of each other, you may select one or both for the step range you've chosen. Select "None" as the type for whichever option you don't want.

To program a contour:

- From the PGM screen, highlight the step below the *last* step in the continuous contour.
- Press the **MORE STEPS** softkey.
- Press the **CONTOUR** softkey.
- You'll notice that "First" and "Last" in the "Step Range" field will be filled in for you.
- If you wish to program an approach, press the **STRAIGHT** or **ARC** softkey as your approach type. Otherwise, press the **NONE** softkey.

  Enter how far from the part you want the approach to begin.
- To program a departure, press the **STRAIGHT** or **ARC** softkey as your departure type. Otherwise, press the **NONE** softkey.

  Enter how far from the part you want your tool to travel.
- If you would like to program a finish cut, enter the amount of material to be removed during the finish cut.

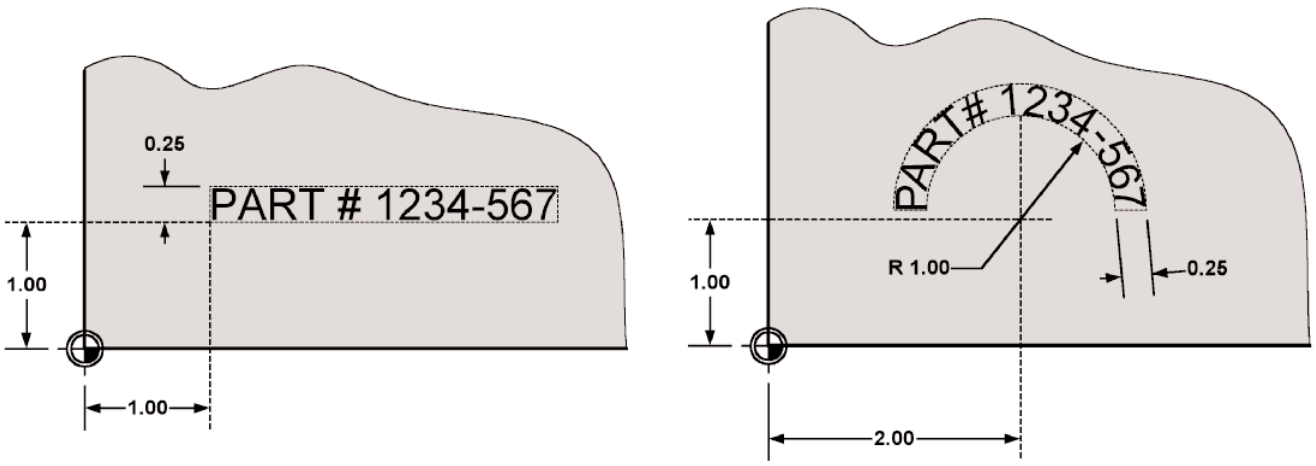
  Enter the feed rate.

  Using the softkeys, select the finish pass direction (**FORWARD** or **REVERSE**).
- Press the **USE** key.

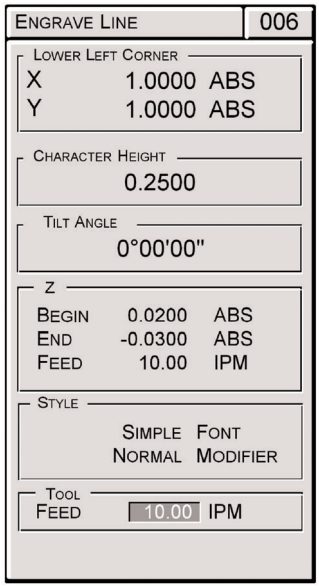
Engrave

With **MILL*PWR***, you have the ability to engrave letters, numbers and symbols, along a straight line or on an arc. Choose from a simple, stick or stencil font. The character height, font and modifier settings you select will define your engraving's appearance.

Condition: The tool diameter being used establishes the spacing between letters.



Engrave Line



To engrave along a line:

- From the PGM screen, press the **MORE STEPS** softkey.
- Press the **MORE STEPS** softkey again.
- Press the **ENGRAVE LINE** softkey.
- Enter the X- and Y-axes coordinates for the point at the lower left corner of your engraving.
- Enter the character height.
- Enter the tilt angle (if any).
- Enter the begin and end depths for the Z-axis.
- Enter the Z-axis feed rate.

- Highlight "Font." If you want to change the font, press the **ENGRAVER FONTS** softkey, highlight the font you want, then press **ENTER**.

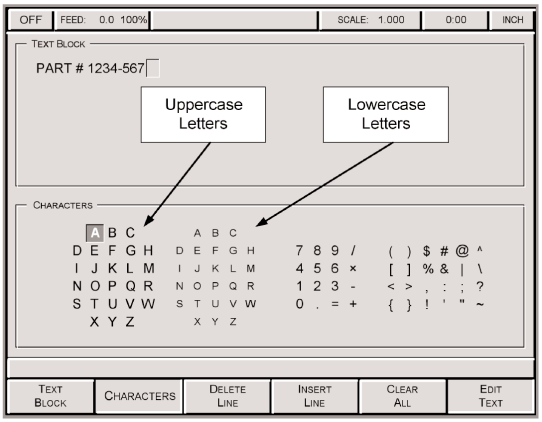
- Highlight "Modifier" and press either the **NORMAL** or **MIRRORED** softkey. "Normal" means that the engraving will be readable from left to right; "Mirrored" will make the engraving appear backwards.

Use "mirrored" if you're making a mold.

- Enter the table's feed rate.

- Press the **EDIT TEXT** softkey.

- Notice that the screen is divided into two sections— "Text Block" and "Characters." You can switch from one to the other by pressing either the **TEXT BLOCK** or **CHARACTER** softkey. Use the arrow keys to move around within each section.



- In the "Character" section, highlight a letter, number, symbol or space. Press the **ENTER** key to make a selection. (The alphabet block on the left side represents capital letters; the block on the right side represents lower case letters.)

  To add a space, move your cursor to a blank spot anywhere within the "Characters" box and press the **ENTER** key.

  You can edit text by pressing the **TEXT BLOCK** softkey. Highlight the character you want to edit, and then press one of the following:

  The **CLEAR** key will delete a character.

  The **DELETE LINE** softkey will erase the entire line of text that the cursor is in and move all following text lines up.

  The **INSERT LINE** softkey will insert a blank text line, moving existing text lines down.

  The **CLEAR ALL** softkey will erase all of the text within the text block.

- When you've finished creating/editing your text, press the **EDIT TEXT** softkey (or press the **USE** key) to return to the "Engrave" screen. **MILL*PWR*** will display the text that you've created on the right.

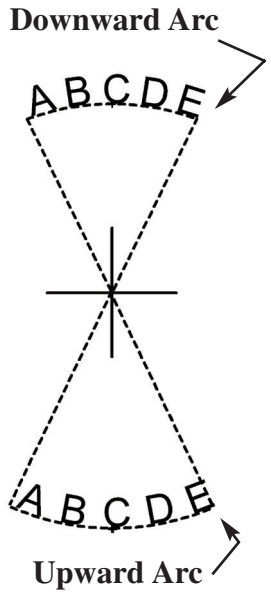
- Press the **EDIT TEXT** softkey if you want to make changes, or press the **USE** key to add the engraving step to your program.

- Press the **EDIT TEXT** softkey when you've finished entering your characters.

- Press the **USE** key.

Engrave Arc

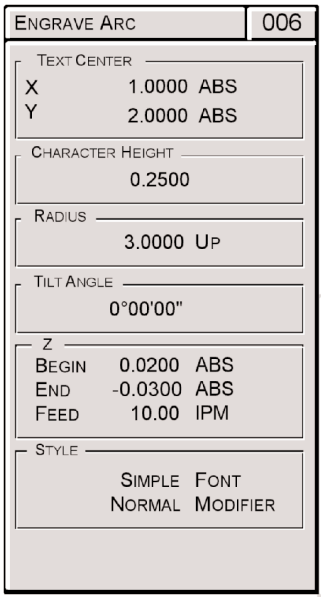
To engrave along an arc:

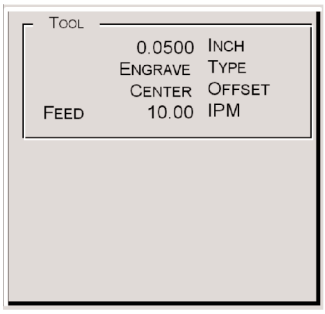


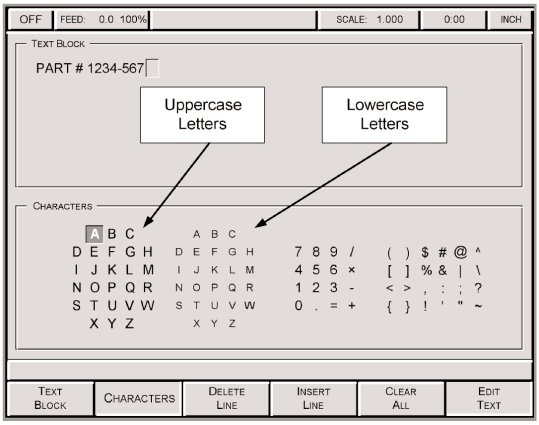
- From the PGM screen, press the **MORE STEPS** softkey.
- Press the **MORE STEPS** softkey again.
- Press the **ENGRAVE ARC** softkey.
- Enter the X- and Y-axes coordinates for the center point of your engraving's arc.
- Enter the character height.
- Enter the radius.
- Select either the **UP** or **DOWN** softkey. "Up" means that the arc will curve upward; "Down" means that the arc will curve downward.
- Enter the tilt angle (if any).
- Enter the begin and end depths for the Z-axis.
- Enter the Z-axis feed rate.
- Highlight "Font." Press the **ENGRAVER FONTS** softkey, highlight the font you want, then press **ENTER**.

Use "mirrored" if you're making a mold.

- Highlight "Modifier." Press either the **NORMAL** or **MIRRORED** softkey. "Normal" means the type will be readable from left to right; "Mirrored" will make the engraving appear backwards.
- Enter the table's feed rate.
- Press the **EDIT TEXT** softkey.






OFF
FEED: 0.0 100%
SCALE: 1.000
0:00
INCH
TEXT BLOCK
PART # 1234-567
Uppercase Letters
Lowercase Letters
CHARACTERS
A B C D E F G H I J K L M N O P Q R S T U V W X Y Z
a b c d e f g h i j k l m n o p q r s t u v w x y z
7 8 9 / 4 5 6 × 1 2 3 - 0 . = +
( ) $ # @ ^ [ ] % & | \ < > , : ; ? { } ! ' " ~
TEXT BLOCK
CHARACTERS
DELETE LINE
INSERT LINE
CLEAR ALL
EDIT TEXT

- Notice that the screen is divided into two sections—"Text Block" and "Characters." You can switch from one to the other by pressing either the **TEXT BLOCK** or **CHARACTER** softkey. Use the arrow keys to move around within each section.

- In the "Character" section, highlight a letter, number, symbol or space. Press the **ENTER** key to make a selection. (The alphabet block on the left side represents capital letters; the block on the right side represents lowercase letters.)

  To add a space, move your cursor to a blank spot anywhere within the "Characters" box and press the **ENTER** key.

  You can edit text by pressing the **TEXT BLOCK** softkey. Highlight the character you want to edit, and then press one of the following:

  The **CLEAR** key will delete a character.

  The **DELETE LINE** softkey will erase the entire line of text that the cursor is in and move all following text lines up.

  The **INSERT LINE** softkey will insert a blank text line, moving existing text lines down.

  The **CLEAR ALL** softkey will erase all of the text in the text block.

- When you've finished creating/editing your text, press the **EDIT TEXT** softkey (or press the **USE** key) to return to the "Engrave" screen. **MILL*PWR*** will display the text that you've created on the right.

- Press the **EDIT TEXT** softkey if you want to make changes, or press the **USE** key to add the engraving step to your program.

- Highlight a letter, number or symbol. Press the **ENTER** key after each selection. (The alphabet block on the left represents capital letters; the block on the right represents lowercase letters.)

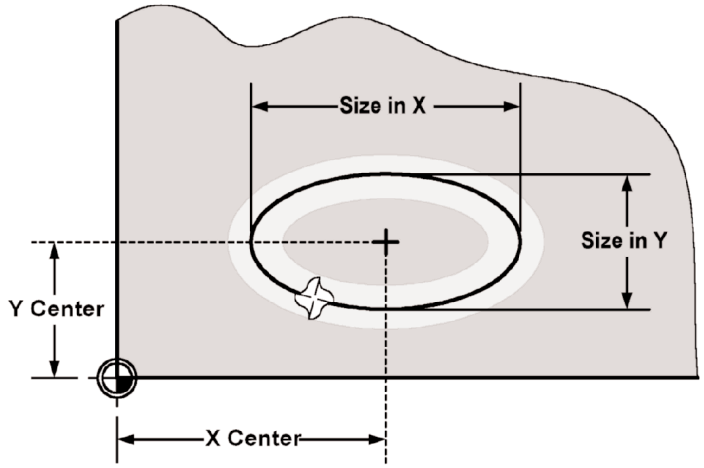
To add a space, move your cursor to a blank spot anywhere within the "Characters" box and press the **ENTER** key.

- Press the **EDIT TEXT** softkey when entering your characters.
- Press the **USE** key.

Ellipse Frame

An ellipse frame is a closed curve with an oval shape. The only offset available for this is center offset. Due to the nature of an ellipse, when the center of the tool tracks an ellipse, its edge will have some error. To minimize the amount of error and cut the most accurate possible ellipse, use the smallest tool diameter available.

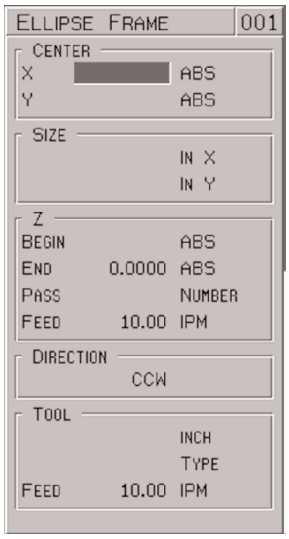
To cut an inside ellipse, subtract the tool diameter from the size in both X-and Y-axes.



To cut and outside ellipse, add the tool diameter to the size in both X-and Y-axes.

To program an ellipsed frame:

- From the PGM screen, press the **MORE STEPS** softkey.
- Press the **MORE STEPS** softkey again.
- Press the **ELLIPSE FRAME** softkey.
- Enter the location of the center point for the X-and Y-axes.
- Enter the size along the X-and Y-axes.
- Enter the depth.
- Select the cutting path direction. Press the **CW** softkey for a clockwise direction or the **CCW** softkey for a counter-clock wise direction.
- Enter the feed rate.



- If you need to program a tilt angle press the **MORE** softkey.

  Tilt Angle
  Highlight "Tilt Angle" and enter the angle (from the X-axis).

- Press the USE key.

Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running your program (refer to **Programming a Tool Step**).

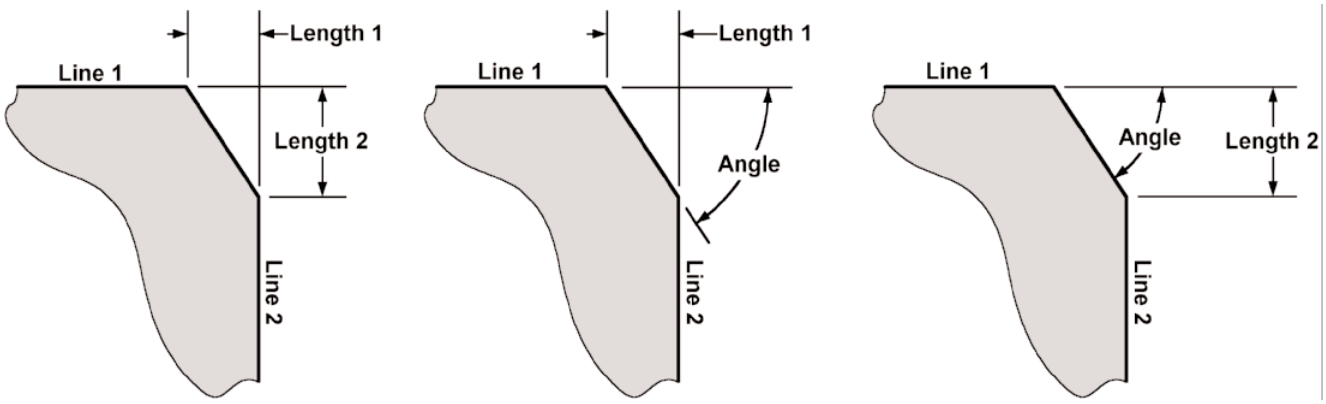
Chamfer

A chamfer is a bevel or line that's inserted between two lines to relieve sharp angles or corners on a part.

You can insert a chamfer between two intersecting lines whose steps are adjacent in the program step.

There are three ways you can program a chamfer:

- With two lengths
- With Length 1 and an angle
- With Length 2 and an angle

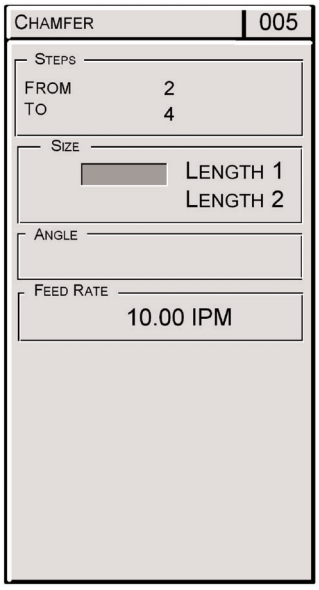


Choose a method based upon information from your print.

You may also close a contour with a chamfer (e.g., a triangle) by inserting the chamfer step immediately after the last step in the contour.

To program a chamfer:

- From the PGM screen, locate the lines you want to insert a chamfer between. Highlight the second line.
- Press the **MORE STEPS** softkey.



- Press the **MORE STEPS** softkey again.
- Press the **CHAMFER** softkey.
- **MILL*PWR*** will automatically fill in the "From" and “To” fields in “Steps” for you. (Press the **CLOSE CONTOUR** softkey if you want to chamfer the end of a contour with the beginning.)
- Enter the distance from the common point of both lines. ("Length 1" refers to the line identified in the "From" field; "Length 2" refers to the line identified in the "To" field.) If you plan to use an angle, enter only one length.
- If you entered one length above, highlight the "Angle" field and enter the chamfer's angle from the X-axis; otherwise, leave it blank.
- Adjust the feed rate, if necessary.
- Press the **USE** key.

Reference Point

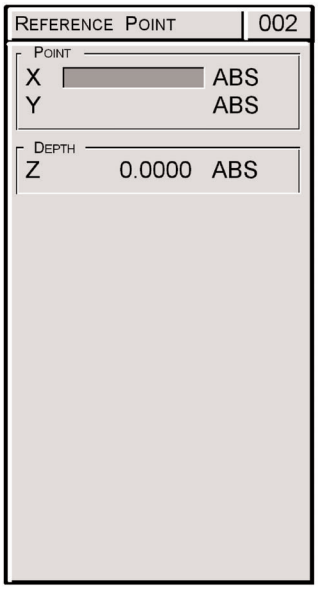
Depicted as a plus sign (+) on your PGM screen, a reference point is a graphical representation of a coordinate in your program.

Reference points are commonly used to identify center points, tangent points and other part features. They can even be used as the basis for incremental moves.

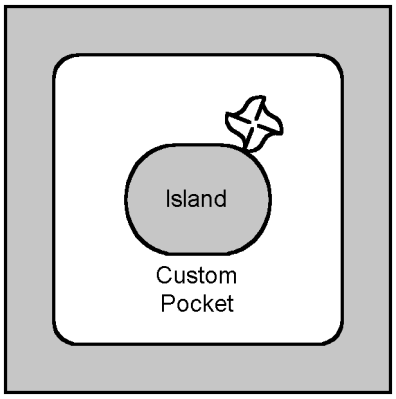
As you program, note that placing a reference point in a continuous tool path will break the path. Otherwise, reference points do not affect your program's performance in any way—in fact, **MILL*PWR*** will skip over them altogether when you run a program.

To program a reference point:

- From the PGM screen, press the **MORE STEPS** softkey.
- Press the **MORE STEPS** softkey two more times.
- Press the **REFERENCE POINT** softkey.
- Enter your reference point's position for X, Y and Z.
- Press the **USE** key.



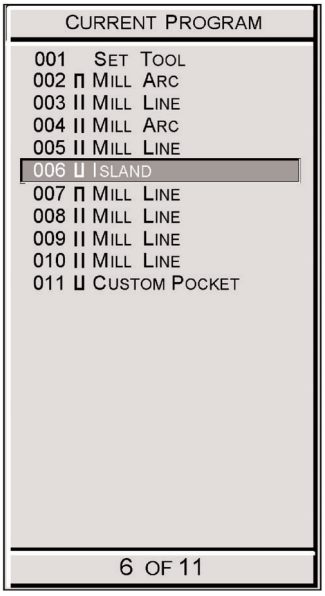
Island



An island is a raised area within a custom pocket that remains after material has been removed from around all of its sides.

Though islands are easy to program, they must be placed correctly within the program sequence. Steps for the island's continuous tool path must appear first, followed by the island step. Steps for the custom pocket's continuous tool path must appear next, followed by the custom pocket step.

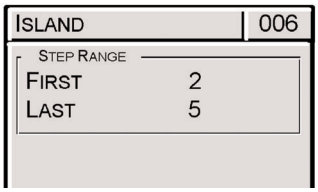
You may program more than one island within the custom pocket.



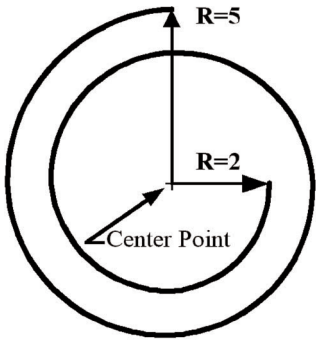
To program an island:

- Program a continuous tool path for the island.
- From the PGM screen, place the cursor below the last step of the island's continuous tool path.
- Press the **MORE STEPS** softkey.
- Press the **MORE STEPS softkey** two more times.
- Press the **ISLAND** softkey.
- Check that the first and last steps listed in the step range match the first and last steps for the island's continuous tool path.

  If they're correct, press the **USE** key; if they're not, press the **CANCEL** key and check that the continuous tool path for the island is correct and/or the correct step has been highlighted.



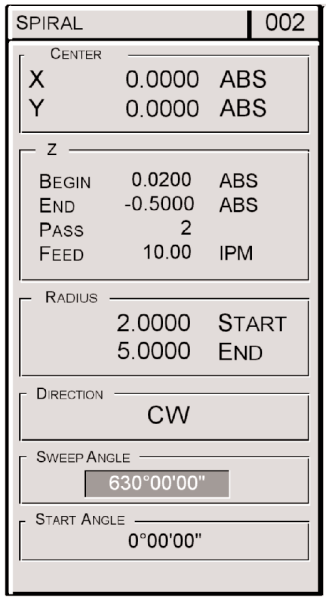
Spiral



A spiral is a winding and gradually widening curve or coil. Spirals are defined by their center point, beginning and ending radii, and sweep angle.

The center point is the (X, Y) coordinate at the core of the spiral. The beginning radius is the distance from the spiral's center point to its starting point, and the ending radius is the distance from the spiral's center point to its ending point.

To program a spiral:

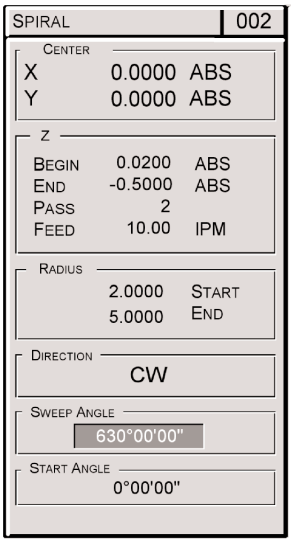


- From the PGM screen, press the **MORE STEPS** softkey.
- Press the **MORE STEPS** softkey two more times.
- Press the **SPIRAL** softkey.
- Enter the spiral's center point.
- Enter the begin and end depths for the Z-axis.
- Enter either the number of passes OR the distance between each pass. “Pass” refers to the cuts that are used to machine the pocket to its “End” depth.
- Enter the Z-axis feed rate.
- Enter a start radius and an end radius (measured from the center point). If the ending radius is less than the start radius, then the tool will cut from the outside in, or vice versa.
- Select the direction by pressing the **CW** softkey for a clockwise direction or the **CCW** softkey for a counter-clockwise direction.

As you're programming, keep in mind that you may "teach" the positions for the center point and depth.

- Enter the sweep angle (e.g., 180 for a half-rotation; 360 for one rotation; 1080 for three rotations, etc.).
- Using the softkeys, choose a tool offset—**LEFT**,**CENTER**, **RIGHT**, **INSIDE** or **OUTSIDE**.
- Enter a table's feed rate.
- If you need to program a start angle or would like to program a finish cut:

Use the calculator keys to let MILLPWR factor the angles for you. For example, enter "3.5 x 360" to create a spiral with three and a half rotations.



Start Angle

You'll want to program a start angle if the beginning point of your spiral on your print is tilted—that is, not positioned at a 0 degree angle (3 o'clock).

Highlight "Start Angle," then enter the angle of the spiral's starting position (measured from the X-axis).

Finish

Enter the amount of material to be removed during the finish cut.

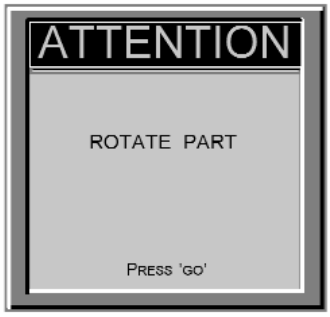
Enter the feed rate for the finish cut.

Select the direction for the finish cut. Press the **CW** softkey for clockwise or the **CCW** softkey for a counter-clockwise direction.

- Press the **USE** key.

Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running your program (refer to **Programming a Tool Step**).

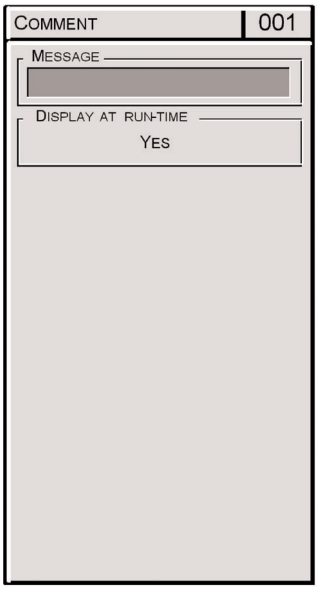
Comment Step



With **MILL*PWR***, you have the ability to insert messages anywhere within a program. These messages can be displayed during machining (at run-time) or as Operator Intervention Messages (OIM). These messages become operational steps within the program and communicate pertinent information—like "ROTATE PART" or "ACTIVATE COOLANT".

For comments that don't require an operator intervention, select "No" when asked if you want the comment displayed at run-time, and **MILL*PWR*** will skip over them during machining. (You can always retrieve the message by highlighting the comment step in your program steps list and pressing **ENTER**.)

To program a comment step:



- From the PGM screen, press the **MORE STEPS** softkey.
- Press the **MORE STEPS** softkey two more times.
- Press the **COMMENT** softkey.
- Enter your message. You may include up to 20 characters, mixing numbers, letters, spaces and symbols if you wish. Press the **ALPHABET** softkey to chose alphabet character needed.

  To add a space, highlight a blank spot within the alphabet box and press the **ENTER** key.

  To add numbers, math symbols (+, -, x, ÷) or a decimal point, use the keypad on the operator console.
- Press the **ALPHABET** softkey again when you've finished entering your message, and the alphabet box will disappear.
- Highlight "Display At Run-time." Press the **YES** softkey to display the message during machining or the **NO** softkey if you don't want the message displayed.
- Press the **USE** key.

Auxiliary Function (AMI Option)

If you purchased an optional AMI (Auxiliary Machine Interface) from ACU-RITE, you can program coolant pumps, automatic lubrication systems and other devices to turn on, off or pulse automatically. Simply insert an “Auxiliary Function” step into your program each time you want to change a device’s status.

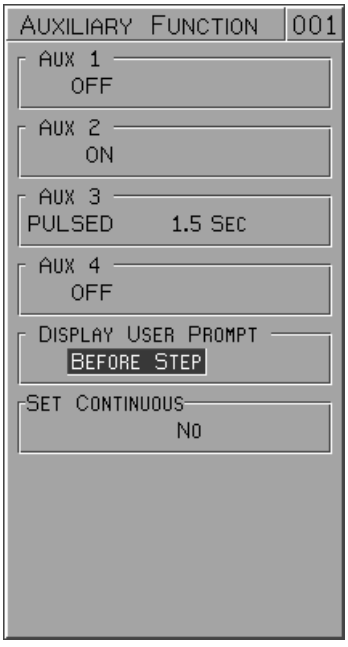
Note: Before you can use the Auxiliary Function feature, the AMI and the output devices you’re controlling must be properly installed (refer to the **AMI Reference Manual** for instructions).

Pausing your program won't affect the auxiliary functions. Each device will continue to run normally. However, **MILL*PWR*** will shut off all of the relays automatically when the program ends. To shut off a device earlier in the program, insert another auxiliary step with the appropriate relay turned "off."

Devices programmed to switch "on" will run continuously as you machine; devices set to "pulse” will run for the length of time (up to 999.9 seconds) you define.

To add an auxiliary function step to your program:

- Check that the AMI is properly connected to **MILL*PWR*** and that each device is connected to the AMI’s J18 output connector (refer to the AMI Reference Manual).
- From the PGM screen, press the **MORE STEPS** softkey.
- Press the **MORE STEPS** softkey two more times.



- Press the **AUXILIARY FUNCTION** softkey.

- Highlight the auxiliary relay(s) that you want to program. (For example, "AUX 1" refers to the output device that's connected to pins 1 and 2 on the J18 connection.) Select from the **OFF**, **ON** and **PULSED** softkeys for the relay(s).

  ON: When the Auxiliary Function Step is executed, an output signal is generated. The signal will continue until another Auxiliary Function Step programmed to turn the signal OFF is executed , the **TABLE STOP** button is pressed, an E-Stop input signal is detected, the program is stopped, or the program ends.

  OFF: When the Auxiliary Function Step is executed, any current signal is turned off.

  Pulsed: When the Auxiliary Function Step is executed, an output signal is generated for the time duration specified. The signal will continue until the time duration ends, another Auxiliary Function Step programmed to turn the signal OFF is executed, the **TABLE STOP** button is pressed, an E-Stop input signal is detected, the program is stopped, or the program ends.

- If you selected "Pulsed," then enter the amount of time (in seconds) that you want the relay to remain on. The maximum amount of time that a relay can pulse is 999.9 seconds.

- Highlight the Display User Prompt field.

  NO: If you do not want an operator prompt to be displayed press **NO**.

  Before Step: Pressing Before Step causes an operator prompt to display before the auxiliary function(s) execute. The prompt will show you what each out put will do. Pressing **GO** will execute the auxiliary function(s).

  After Step: Pressing After Step causes the operator prompt to display after the auxiliary function(s) execute. The program will pause until **GO** is pressed or a signal from an auxiliary device is received.

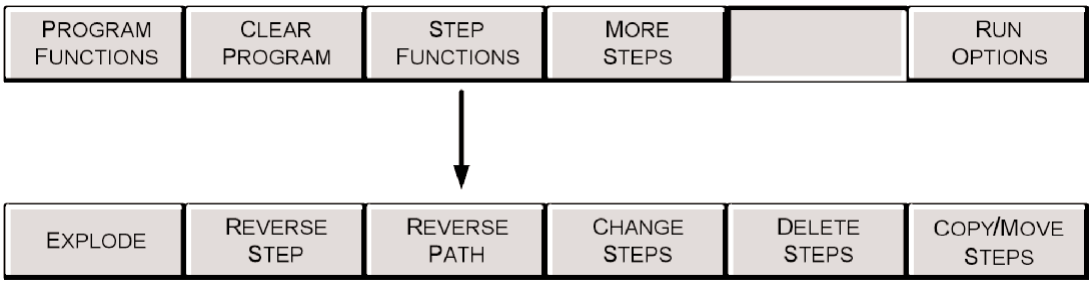
- Pressing "Yes" for the Set Continuous field allows an Auxiliary Function Step to be entered (e.g. activate coolant) without interrupting the continuous milling path.

- Press the **USE** key.

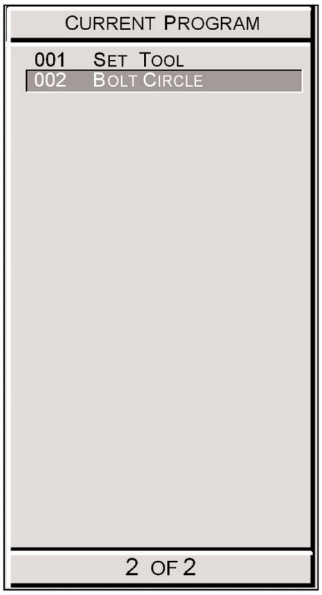
  Note: Check that the devices you are using are powered on before running your program.

Step Functions Softkey

Additional functions are available from the PGM screen by pressing the **STEP FUNCTIONS** softkey.



Explode

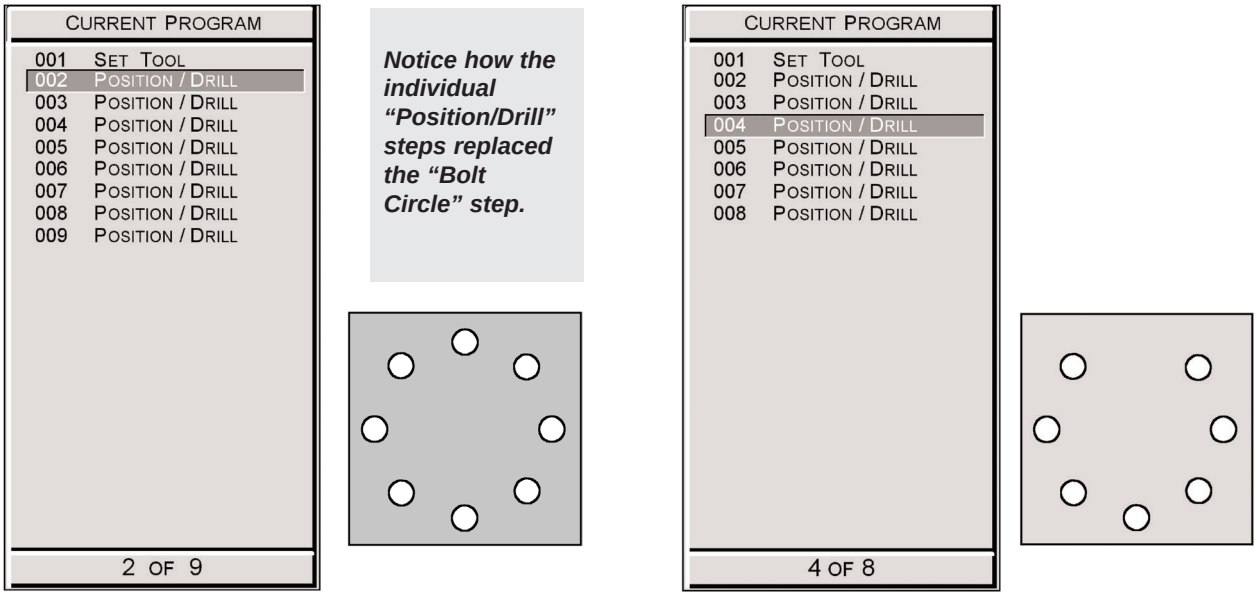


This feature will "explode" a program step into several, more detailed steps. You can explode the following functions:

- All "Holes" functions (row, frame, array and bolt circle)
- Repeat, Mirror and Rotate
- Engrave

In the example on the right, a bolthole circle with eight holes has been programmed. Now use the same basic pattern, but with seven holes.

To edit the program, first highlight the “Bolt Circle” step and then press the **EXPLODE** softkey.


CURRENT PROGRAM
001 SET TOOL
002 POSITION / DRILL
003 POSITION / DRILL
004 POSITION / DRILL
005 POSITION / DRILL
006 POSITION / DRILL
007 POSITION / DRILL
008 POSITION / DRILL
009 POSITION / DRILL
2 OF 9
Notice how the individual "Position/Drill" steps replaced the "Bolt Circle" step.
CURRENT PROGRAM
001 SET TOOL
002 POSITION / DRILL
003 POSITION / DRILL
004 POSITION / DRILL
005 POSITION / DRILL
006 POSITION / DRILL
007 POSITION / DRILL
008 POSITION / DRILL
4 OF 8

MILL*PWR* will explode the "Bolt Circle" steps into eight steps (shown on the left, above).

Next highlight the step that represents the hole that needs to be deleted and press the **CLEAR** key. In the graphic above, chose to delete the hole programmed in step 4.

As you can see in the graphic on the right, the bolthole circle pattern now has seven holes.

To explode a step:

- Highlight the step.
- Press the **STEP FUNCTIONS** softkey.
- Press the **EXPLODE** softkey.

The step that you exploded will be replaced with the individual lines, arcs or positions that made up the step.

Reverse Step

The reverse step option instantly switches the "From" and "To" points and tool offset.

To reverse a milling function:

- From the PGM screen, use the arrow keys to highlight the step that you want to reverse.
- Press the **STEP FUNCTIONS** softkey.
- Press the **REVERSE STEP** softkey.

Reverse Path

With the reverse path option, you can reverse any continuous tool path. This will especially come in handy when you're working with DXF files. As you import these types of files, **MILL*PWR*** will sort and then group the steps into a logical order, creating continuous paths. In some cases, the paths may need to be reversed after they've been imported so that your tool's offset, direction and beginning and end points satisfy your machining requirements.

As you become more familiar with this feature, you'll find other creative ways to use it to your advantage. For instance, you can save time as you're cutting a part by using a heavy cutting tool and a conventional cut for a rough first pass. On the second pass, switch to a finish cutter, then copy and reverse the path for a climb cut on the finish pass.

To reverse a continuous tool path:

- From the PGM screen, highlight any step within the continuous tool path that you want to reverse.
- Press the **STEP FUNCTIONS** softkey.
- Press the **REVERSE PATH** softkey.

Change Steps

The "Change Steps" feature gives you the ability to change or edit the depth, offset and feed rate of several steps simultaneously.

Note: You can use this feature from anywhere within your program—you don't need to highlight a specific step within the step range.

To use the “Change Steps” feature:

- From the PGM screen, press the **STEP FUNCTIONS** softkey.
- Press the **CHANGE STEPS** softkey.
- Enter the first and last step numbers that you would like to change in the “Step Range” field.
- Highlight each field that you want to change and enter the new data.
- Press the **USE** key.

Note: It’s a beneficial to highlight the changed steps in the program sequence, then press the **ENTER** key. Each step should include the new settings.

Delete Steps

MILL*PWR* gives you the option of deleting steps in two ways: using the **DELETE STEPS** softkey or using the **CLEAR** key.

When you're deleting single steps, we suggest highlighting the step then pressing the **CLEAR** key.

When you're deleting a range of steps—after you've merged programs, for example—the "Delete Steps" feature is usually your best option.

To delete a group of steps from your program:

- From the PGM screen, press the **STEP FUNCTIONS** softkey.
- Press the **DELETE STEPS** softkey.
- In the “Step Range” field, enter the first and last step numbers that you would like to delete.
- Press the **USE** key.

Copy/Move Steps

Copy/Move operations make it easy for you to duplicate or rearrange steps within your program. You'll find the "Move" feature especially useful for editing steps generated from a DXF file.

After you press the **COPY/MOVE STEPS** softkey, you'll be asked to enter a step range, then either copy or move the steps.

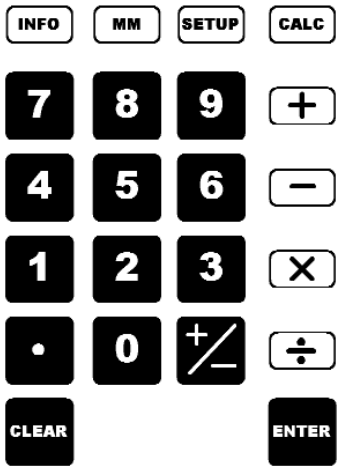
Press the **MOVE** softkey to relocate the steps. Press the **COPY** softkey to create an identical copy of the steps you've chosen (the original steps will remain in place). After you press the **USE** key, the new steps will be inserted into your program.

To copy or move steps:

- From the PGM screen, highlight a step where you would like to add or insert the step(s) that you want to move or copy.
- Press the **STEP FUNCTIONS** softkey.
- Enter the first and last step numbers that you would like to move or copy.
- Press the **COPY/MOVE STEPS** softkey.
- Press either the **MOVE** or **COPY** softkey.
- Press the **USE** key.

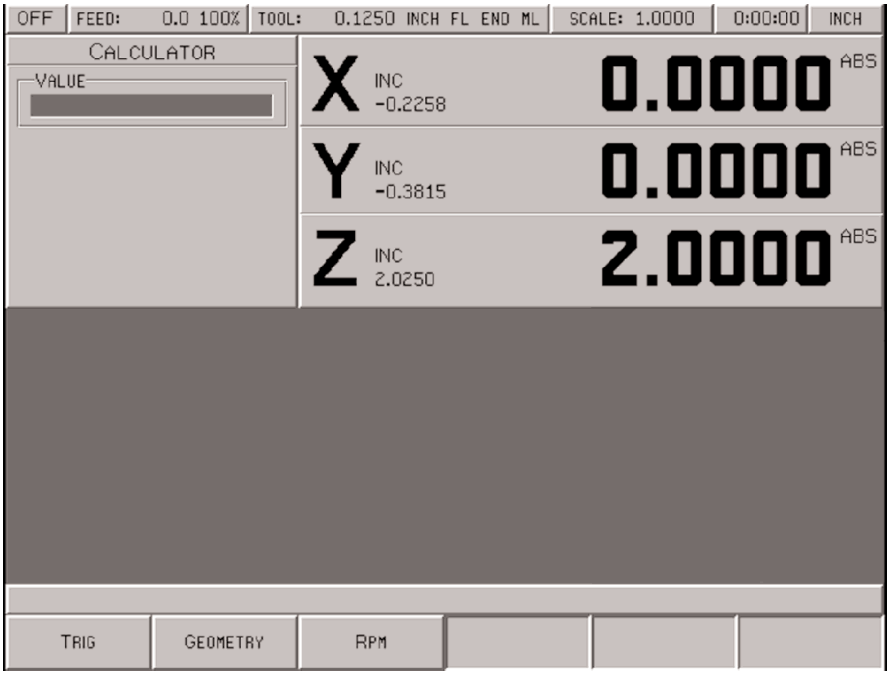
This page intentionally left blank.

Calculator



MILL*PWR*'s built-in calculator is capable of handling everything from simple arithmetic to complex trigonometry, geometry and RPM calculations. The numeric keypad resembles a standard calculator with keys for numbers 0 through 9, four math function symbols (+, - , x, and ÷), a decimal point, and a positive/negative sign (+/-).

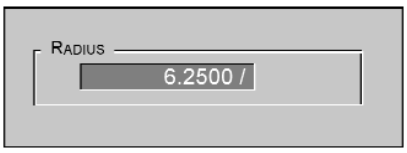
The calculator is accessible from nearly any screen or field. Press the **CALC** key to access the "stand-alone" calculator and the **TRIG**, **GEOMETRY** and **RPM** softkeys. Calculations can be entered directly into a highlighted field.



Math Functions (+, -, x, ÷)

Math functions may be performed separately in the stand-alone calculator or directly in the field you're working in. For example, let's say we need to enter the radius of a circle pocket, but your print only shows the diameter, 6.25. Here's an easy way to figure out the radius:

- Highlight the radius field for the circular pocket.
- Using the keypad, enter the value for the diameter, **6.25**.



- Press the ÷ key.
- Press the **2** key.
- Press the **ENTER** key. The answer, 3.125, will appear in the "Radius" field.

When you need to enter more than one calculation into a numeric field, the calculator will perform multiplication and division before it performs addition and subtraction. If you were to enter 3 + 1 ÷ 8, **MILL*PWR*** will divide one by eight, then add three for an answer of 3.125.

Trig Functions

As we showed above, when you're entering data for a milling or positioning function, your print may not provide you with all of the information you need. If an unknown value can be expressed as a trigonometry function, square root or the square of a number, **MILL*PWR*** can calculate the dimension for you in the trigonometry calculator.

Using the trig calculator:

- While in a numeric field (such as the "To" field for a Line), press the **CALC** key, and then press the **TRIG** softkey. The following softkeys will appear:

SHIFT	COS	SIN	TAN	SQR ROOT	(
	ARC COS	ARC SIN	ARC TAN	SQR	)

- Enter a number and then press the softkey with the appropriate trigonometry math function. (Press the **SHIFT** softkey to switch between the upper and lower functions.)

For example, to enter a radius that has a value equal to the square root of 2, follow this procedure:

Highlight the radius field.

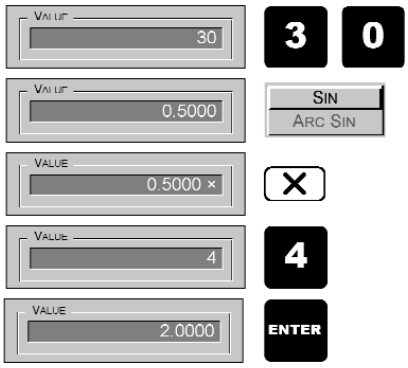
Press the **2** key.

Press the **CALC** key.

Press the **TRIG** softkey.

Press the **SQR ROOT** softkey. A value of 1.4142 will appear.

Whenever you're calculating the SIN, COS or TAN of an angle, enter the angle first and then press the appropriate softkey. If you were to enter a value of 30 into a numeric field, then press the **SIN** softkey, a value of 0.5000 (the SIN of 30 degrees) would appear.



Continuing with the example on the right, let's assume that we are trying to set up a 4" SIN plate at a 30° angle and need to know how tall the block underneath it should be. If we multiply 0.5000 by 4, we know that we need to use a 2" block.

The parentheses softkeys can be used to group calculations within an arithmetic equation. For example,

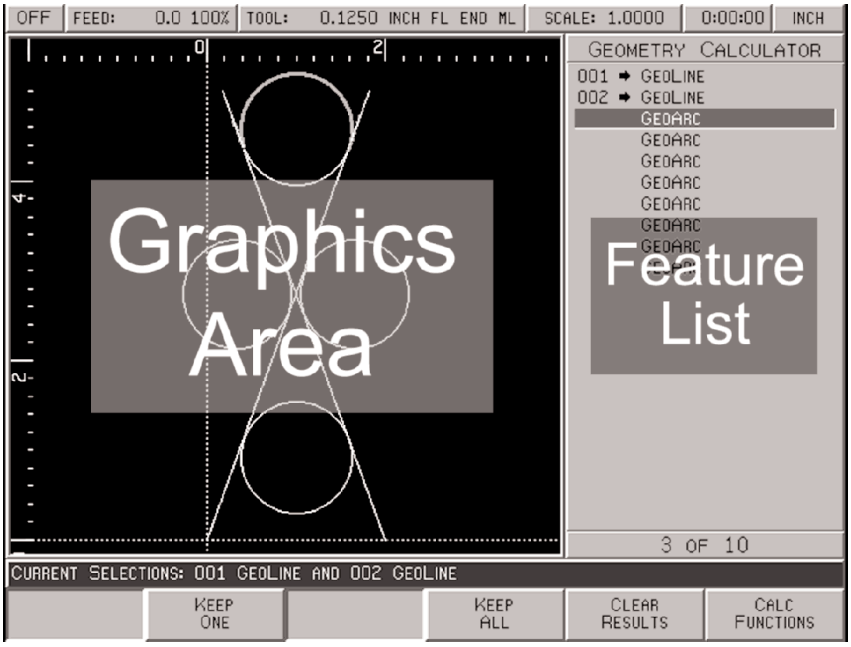
2 + 1 ÷ 16 equals 2.0625, but (2 + 1) ÷ 16 equals 0.1875.

The parentheses are not always displayed during the keystroke sequence, but they are remembered by **MILL*PWR*** as it calculates the correct answer.

Geometry Functions

Working with the Geometry Calculator

To open the geometry calculator (GeoCalc), press the **CALC** key, and then press the "Geometry" softkey.



With **MILL*PWR***'s GeoCalc, you can calculate missing coordinates (such as the tangent point between a line and an arc) using the information that appears on your print.

The kinds of calculations that GeoCalc performs depends on the items you select and whether you're trying to find a point, line, or arc. The table below lists all of the points, lines and arcs that can be found by GeoCalc.

Items	Find Point	Find Line	Find Arc
2 Points	Midpoint between the given points.	Line between the given points.	Given a radius, all arcs through the given points. The Auto Radius softkey gives you two arcs, each tangent to both points with the midpoint as its center point.
1 Point 1 Line	Point on the given line which, with the given point, would form a line perpendicular to the given line.	Lines through the given point, one parallel and one perpendicular to the given line.	Given a radius, all arcs through the given point and tangent to a given line. The Auto Radius softkey gives you two arcs, each tangent to the given line with the given point as its center point.
1 Point 1 Arc	Points which, with the given point, would form lines tangent to the given arc and the shortest line from the given point to the given arc.	Lines through the given point tangent to the given arc, and the shortest line through the given point to the given arc.	Given a radius, all arcs through the given point and tangent to the given arc. The Auto Radius softkey gives you all arcs tangent to the given arc with the given point as its center point.
1 Line	End points of the given line.	Given a distance, lines parallel to the given line; lines perpendicular to the given line through its end points.	
2 Lines	Intersection point of the given lines.	Line that bisects the angle formed where the given lines intersect; line perpendicular to the bisector.	Given a radius, all arcs tangent to both given lines.
1 Line 1 Arc	Points where given line intersects given arc.	Lines perpendicular to the given line and tangent to the given arc; shortest line to the given arc which is perpendicular to the given line.	Given a radius, all arcs tangent to the given line and the given arc.
1 Arc 2 Arcs	Center point, end points.	Given an angle, all lines tangent to and through the center point of the arc; lines from the end points to the center point.	
1 Arc	Intersection points.	Lines tangent to arcs; shortest line between arcs.	Given a radius, all arcs tangent to both arcs.

Using GeoCalc

- Press the **POS**, **LINE** or **ARC** key and enter the information that's on your print. (Refer to **Program Steps** for assistance.)

- To find a line, point or arc, highlight the feature(s) you wish to use, and then press the **SELECT FEATURE** softkey. An arrow will appear beside the feature(s) you've selected. (Only two features can be selected at one time.)

- Press the **FIND POINT**, **FIND LINE** or **FIND ARC** softkey—GeoCalc will automatically find

  all the possible points, lines or arcs for the features you've selected.

- Either highlight the feature you wish to keep and press the **KEEP ONE** softkey OR press the **KEEP ALL** softkey and retain all the features that are on the screen.

The **RETURN FEATURE** softkey transfers the object(s) you've *highlighted* from GeoCalc and into your program or milling function, freeing you from having to write the information down and then entering it in number-by-number from the keypad.

Calculator Functions

GeoCalc has other useful features that make programming easier, like saving and loading calculations. Press the **CALC FUNCTIONS** softkey to access the following functions:

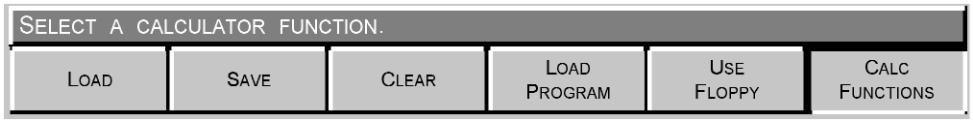
Saving Results Calculated in GeoCalc

GeoCalc lets you save your calculations on **MILL*PWR***'s internal memory or on a floppy diskette so that you can retrieve them at a later time.

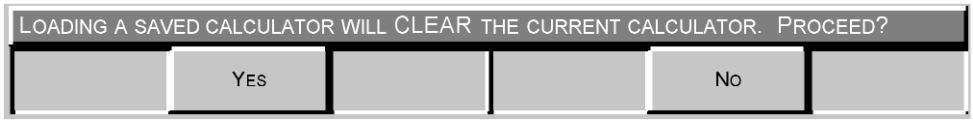
Note: You may only save the results of one GeoCalc session at a time onto **MILL*PWR***'s internal memory or onto a floppy disk. Use one floppy disk for each GeoCalc session you want to save.

To save your GeoCalc results onto **MILL*PWR***'s internal memory:

- From GeoCalc, press the **CALC FUNCTIONS** softkey.



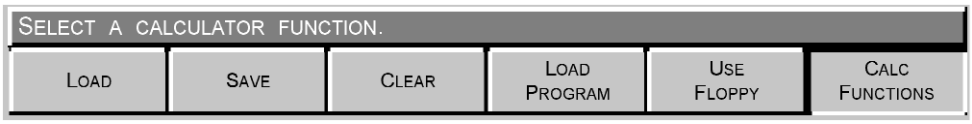
- Press the **SAVE** softkey.



- To replace what you've previously saved, press the **YES** softkey. If you do not wish to proceed, press the **NO** softkey.

To save your GeoCalc results onto a 3 ½" floppy disk:

- From GeoCalc, press the **CALC FUNCTIONS** softkey.
- Insert a floppy disk into the 3 ½" floppy disk drive.
- Press the **USE FLOPPY** softkey.
- Press the **SAVE** softkey.



If you've already saved the results from another GeoCalc session onto the 3½" floppy disk, you will be asked if you want to replace what you previously saved. To save over the older session, press the **YES** softkey. If you don't want to save over the older session, press the **NO** softkey. Insert a new 3½" floppy disk and repeat the procedure.

Loading GeoCalc Results That You Previously Saved

Once you've saved the results from GeoCalc, you can reload them into GeoCalc at any time.

To load GeoCalc results into GeoCalc from **MILLPWR**'s internal memory:

- From GeoCalc, press the **CALC FUNCTIONS** softkey.
- Press the **LOAD** softkey.

To load GeoCalc results into GeoCalc from a 3½" floppy disk:

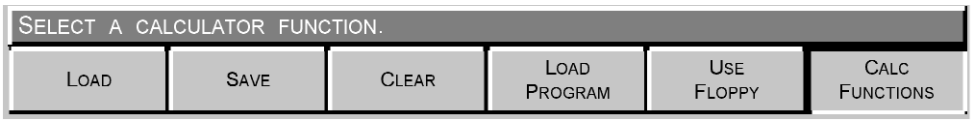
- From GeoCalc, press the **CALC FUNCTIONS** softkey.
- Insert a 3 ½" floppy disk into the floppy disk drive.
- Press the **USE FLOPPY** softkey.
- Press the **LOAD** softkey.

Loading Programs into GeoCalc

You can load your current or open program directly into GeoCalc. Because GeoCalc automatically recreates your program's features, you don't have to reenter information.

To load the current program's features into GeoCalc:

- From GeoCalc, press the **CALC FUNCTIONS** softkey.



- Press the **LOAD PROGRAM** softkey.

Clearing GeoCalc

Once you're finished working in GeoCalc, you can clear the screen.

To clear GeoCalc:

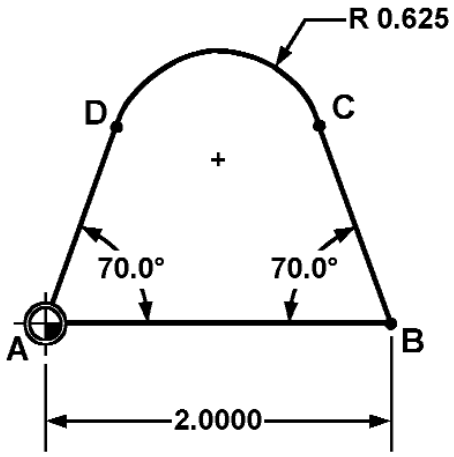
- From GeoCalc, press the **CALC FUNCTIONS** softkey.
- Press the **CLEAR** softkey.
- Press the **YES** softkey to clear the screen. If you do not wish to proceed, press the **NO** softkey.

Example Problem

Occasionally, a print will not provide all of the dimensions needed to program the part. In the illustration below for example, the coordinates of points C and D, where the arc is tangent to the sides, are unknown. The coordinates will have to be identified before lines B-C and A-D and the arc that extends from point D to point C can be programmed.

Strategy

First program the line from A to B directly from the print provided. Then use GeoCalc to calculate the rest.

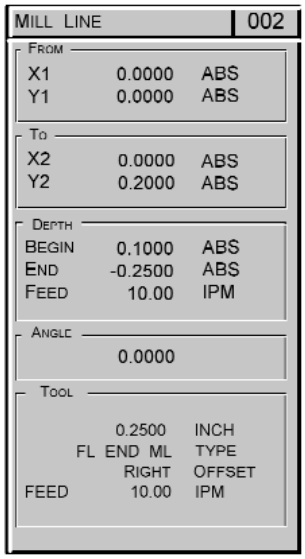


The strategy will be:

- Enter the line from A to D.
- Enter the line from B to C.
- Select lines A-D and B-C, and then use the **FIND ARC** function.
- Calculate points C and D, then return their coordinates and the arc into the program.

Starting the Program

In this example, point A is shown as datum. Its absolute coordinates are 0, 0. In the part program, draw the line by entering the absolute coordinates for point A (0, 0) in the "From" field and point B (2, 0) in the "To" field. Enter the Z-axis and offset information as needed.



- Press **USE** to finish programming the line.

Now use GeoCalc to find the coordinates for points C and D.

Entering the Lines

Press the **CALC** key, and then press the **GEOMETRY** softkey. Now, press the **LINE** key and construct the line from A to D.

- Press the **LINE** key

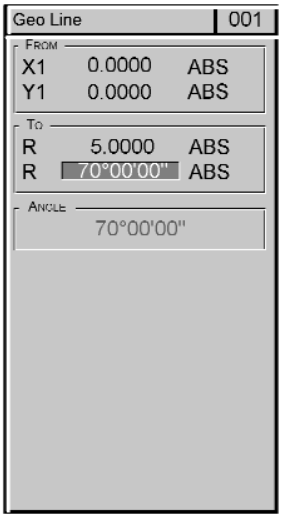
- Enter the coordinates for the "From" point (point A): X = 0, Y = 0

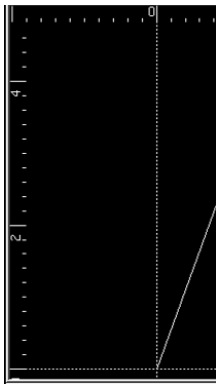
- The polar coordinates are R2 (radius) and A2 (angle). (Remember, a polar coordinate is a position defined by an angle and a radius.)

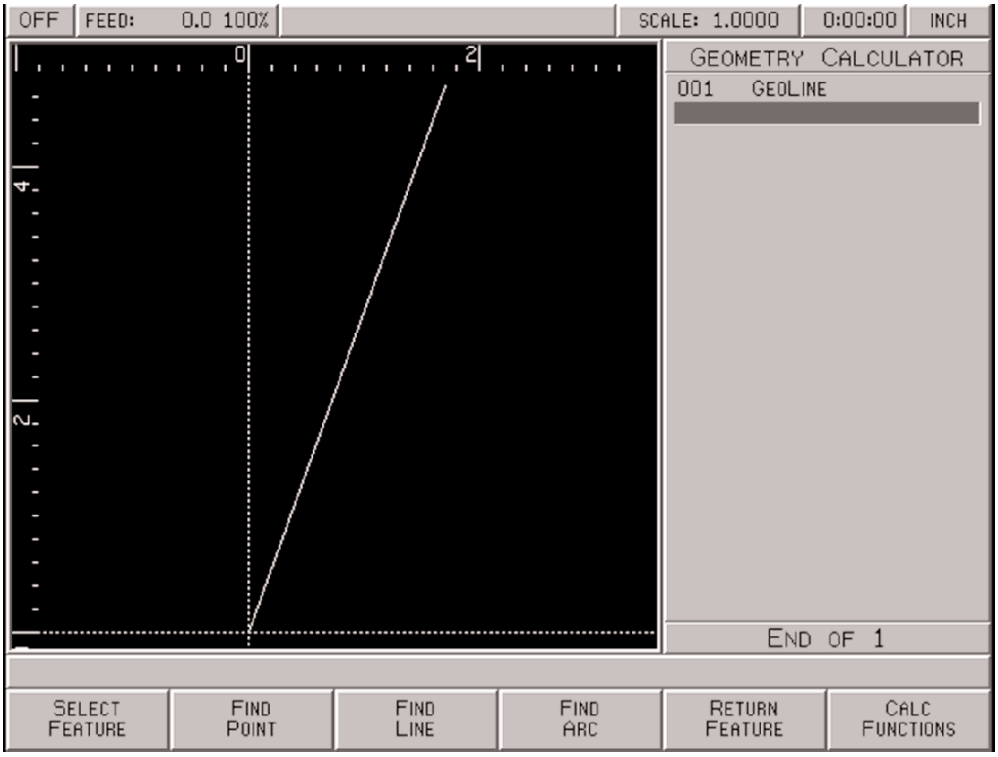
- Since GeoCalc doesn't need the length of the line, enter a radius value of 5".

- The angle of the line from datum is 70°.

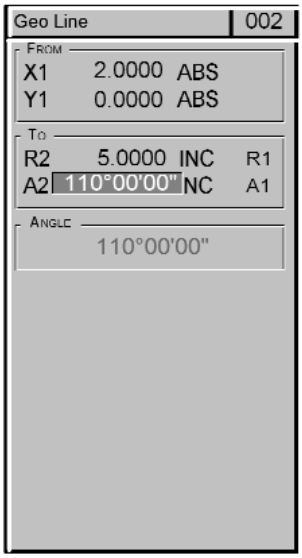
- Press the **USE** key, and a line starting at point A and traveling through point D will appear.







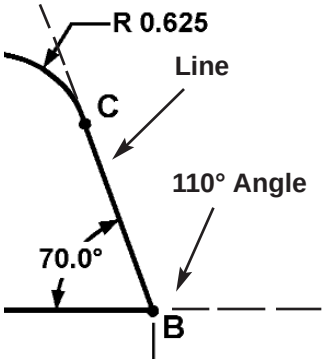
Next, enter the dimensions for the line that reaches from point B through point C.



- Press the **LINE** key.
- In the "From" field, enter point B's coordinates, X1 = 2 and Y1 = 0.

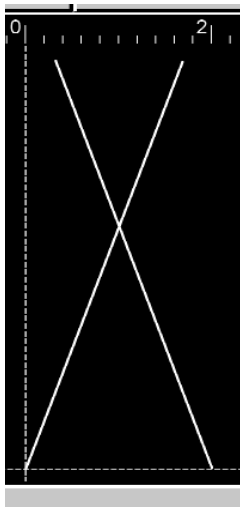
Enter the "To" point. Use polar coordinates, since the angle is known and the radius can be estimated.

- Press the **POLAR** softkey. Estimate "5" for the radius (R2).
- Press the **ABS/INCR** key to switch to incremental measuring (because the measurement is taken from point B, not from datum).
- Highlight the A2 field.



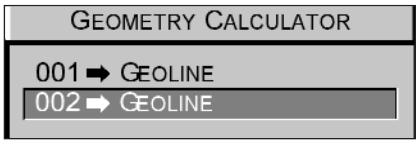
The print shows that the angle is 70°. But if 70° is entered, the line will slant the wrong way. The complementary angle is needed instead. Figure it out by subtracting 70° from 180°, the total angle of the line.

- You can either enter the answer, 110°, directly into the (A2) field, or you can calculate the answer using the keypad ("180-70").
- Since the angle is measured from point B (incremental zero) press the **ABS/INCR** key again.
- Press the **USE** key, and line B-C will appear.



Finding the Arc

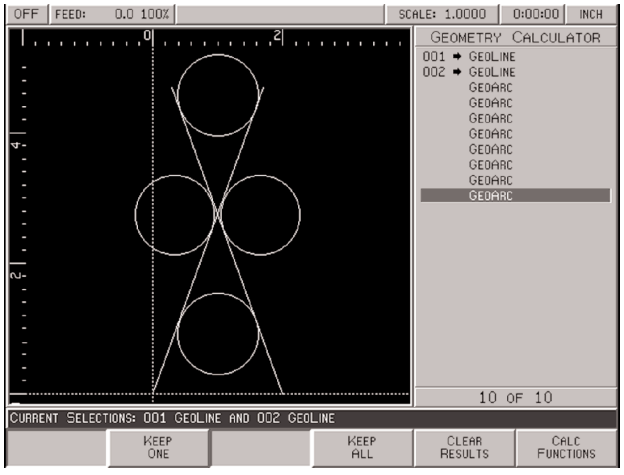
Now that the lines are complete, it's time to find the arc. Do this by selecting the lines that have been drawn and then using GeoCalc to find the arc.

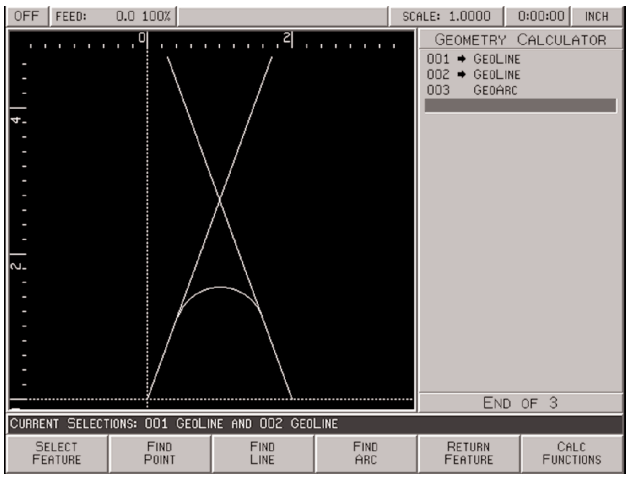


- Highlight 001 GEOLINE and press the **SELECT FEATURE** softkey. The arrow indicates that the feature has been selected.
- Now highlight 002 GEOLINE and press the **SELECT FEATURE** softkey.
- Press the **FIND ARC** softkey. The message bar will display a field for the arc radius.
- The print shows that the arc's radius is 0.625. Using the keypad, enter "0.625," and then press the **USE** key.

GeoCalc will display all of the arcs that have a 0.625 radius and are tangent with the two lines that have been selected (A-D and B-C).

- Using the arrow keys, scroll through the list of found GeoArcs until the arc desired is highlighted.
- Press the **KEEP ONE** softkey, and GeoCalc will save the arc as item 003.
- Arrow down and then press the **CLEAR RESULTS** softkey to erase the remaining arcs.



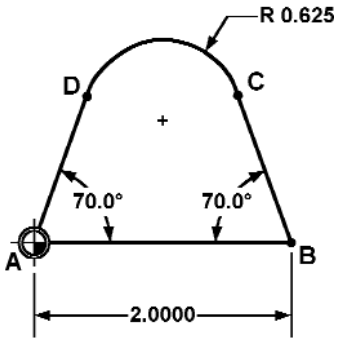


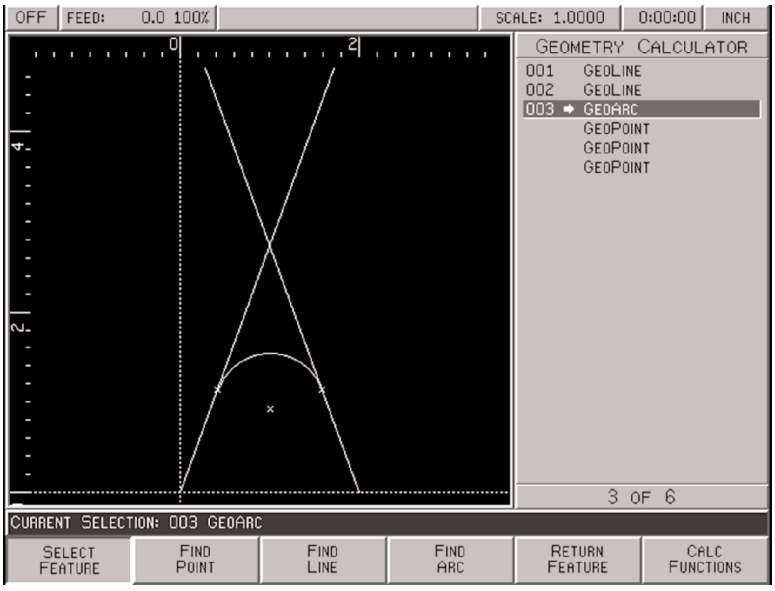
Finding the Points of Tangency

The points of tangency between the lines and the arc (the coordinates for points C and D) need to be determined. Now that the arc is in place, the coordinates of its end points are needed. Use GeoCalc's "Find Point" feature to find the end points of the arc.

- Highlight 001 GEOLINE and press the **SELECT FEATURE** softkey. This will de-select the line and cause the arrow to disappear.
- Highlight 002 GEOLINE and de-select it.
- Highlight 003 GEOARC and press the **SELECT FEATURE** softkey to *select* it.
- Press the **FIND POINT** softkey. Three points will appear—one is the center of the arc and the others are the ends of the arc. The point on the right is the one needed; it's point C on the drawing.
- Using the arrow key, scroll down through the list of "GeoPoints" until the point on the right is highlighted. Press the **KEEP ONE** softkey.

The tangent point (C) is now stored in the GeoCalc as item 004.

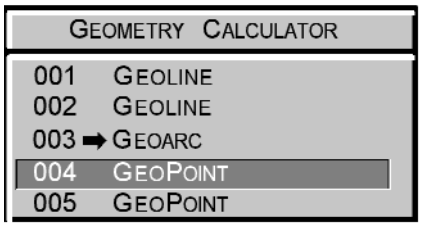




Returning Features

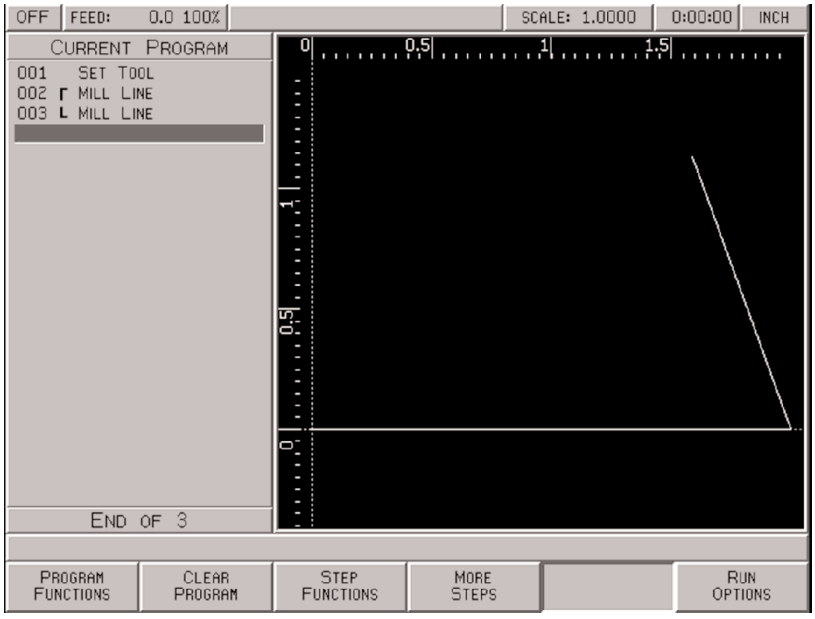
Now bring the GeoCalc results back into the program. When you press the **RETURN FEATURE** softkey, GeoCalc will transfer the feature you've identified into your part program.

- Press the **CANCEL** key to return to the program. Check that your cursor is just *below* the line previously programmed (item 001).
- Press the **LINE** key to begin a new "Mill Line" form. The "From" point is automatically set at point B. The "X2" field should be highlighted. This is point C, one of the points found in GeoCalc.
- Press the **CALC** key, then the **GEOMETRY** softkey.
- Highlight the GeoPoint representing point C (step 004). Press the **RETURN FEATURE** softkey. The program screen will return and point C's coordinates will appear in the "To" field.

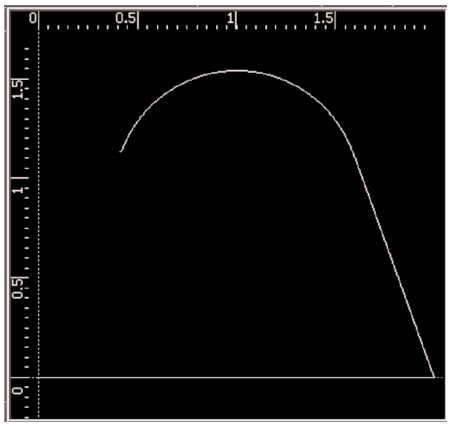


- Press the **USE** key to accept the line into the part program.
- Press the **CALC** key and then the **GEOMETRY** softkey.

The features in the calculator will not be lost until the MILLPWR is turned off or until you clear the calculator.



- Highlight the arc.

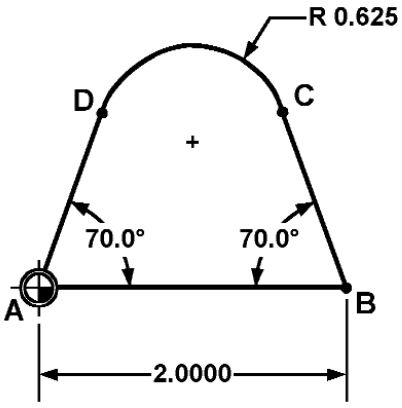


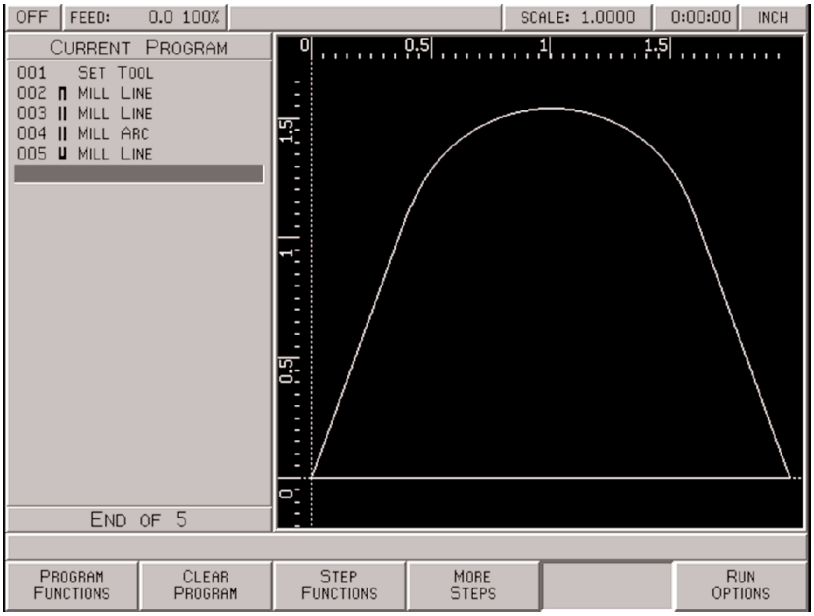
- Press the **RETURN FEATURE** softkey. **MILL*PWR*** will automatically add an arc step to your program and place the arc's information into the form.

- Press the **USE** key to accept the arc step into your part program.

- Press the **LINE** key. This line ends at datum, so enter “X2 = 0” and “Y2 = 0” for the “To” point, and press the **USE** key to keep the line.

The contour is now complete.



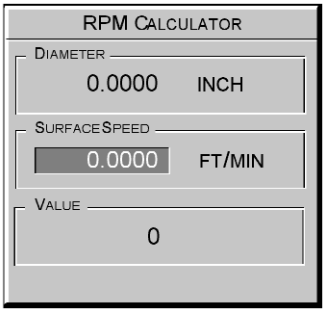


RPM Functions

MILL*PWR* includes a built-in RPM calculator that allows you to make RPM calculations in seconds. It calculates the RPM based upon the tool diameter and surface speed.

If your cursor is in the RPM field of a "Set Tool" step before you open the RPM calculator, **MILL*PWR*** can transfer the result directly into the field for you.

To use the RPM calculator:



- Press the **CALC** key, then the **RPM** softkey.
- Enter the diameter of your tool.
- Enter the surface speed for the material and cutter type.

  Press the **SURFACE SPEED TABLE** softkey to view a list of recommended surface speeds.
- If you need to switch from feet per minute to meters per minute, or vice versa, use the **FEET/MIN** and **METERS/MIN** softkeys.

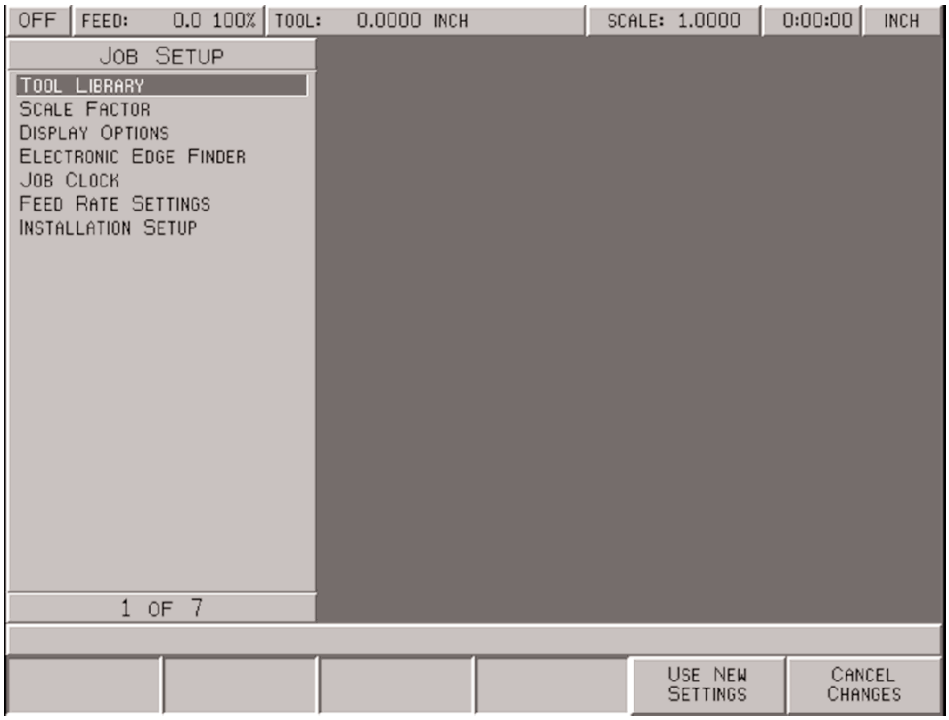
- Press the **ENTER** key. **MILL*PWR*** will calculate the RPM value.
- If your cursor was in the RPM field of a set tool step before you accessed the RPM calculator, press the **USE** key and **MILL*PWR*** will transfer the result directly into the "RPM" field for you.

Setup

Press the **SETUP** key to access the setup parameters. The setup parameters are grouped into two categories: Job Setup and Installation Setup.

Job Setup parameters can easily be changed to accommodate specific machining requirements for each job.

Installation Setup parameters, on the other hand, are established during the initial installation and, with few exceptions, should not be changed. For this reason, the installation setup parameters are protected by a passcode. The passcode is located on the page before the table of contents. You can remove it and store it in a separate location to prevent accidental or unauthorized adjustments to your system.



To change your setup parameters:

- From Setup, highlight a topic, then press **ENTER**. Change the settings, then press the **USE NEW SETTINGS** softkey to activate them. (Or press the **CANCEL CHANGES** softkey to exit Setup and return to the previous screen without making any changes.)

Job Setup

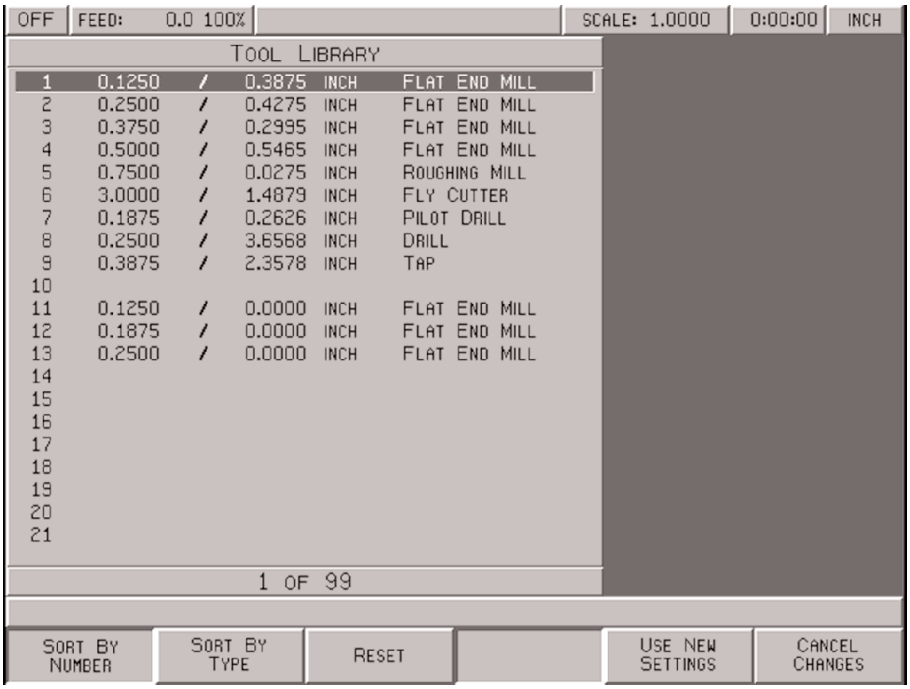
Tool Library

MILL*PWR* contains a tool library that can store the diameter, length, unit of measure, and tool type for up to 99 tools. **MILL*PWR*** only requires that you provide the tool's diameter; however, it's a good idea to provide additional information so that each tool is easier to identify from the tool list.

Set up your tool library so that the tools you use most often appear first.

You can sort your tool library by number or type.

> ⚠ Pressing the **RESET** softkey will clear all of the tools that are programmed in the tool library. Only use this key when you want to start over.

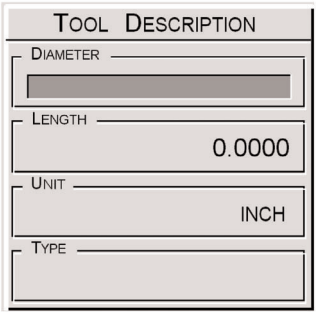


Using the tool library without repeatable tool lengths:

To enter a tool into the Tool Library:

- From Setup, highlight “Tool Library” then press **ENTER**.
- Highlight a blank row in the Tool Library and press **ENTER**.
- Enter the tool’s diameter.
- Highlight the “Type” field.
- Press the **TOOL TYPES** softkey.
- Highlight the tool type that describes the tool you’re using. Press **ENTER**.
- Press the **USE** key.
- Continue with Setup or press the **USE NEW SETTINGS** softkey.

If you need to, change the units (inches or mm) for the tool you’re using before you set the length.



To edit an existing tool:

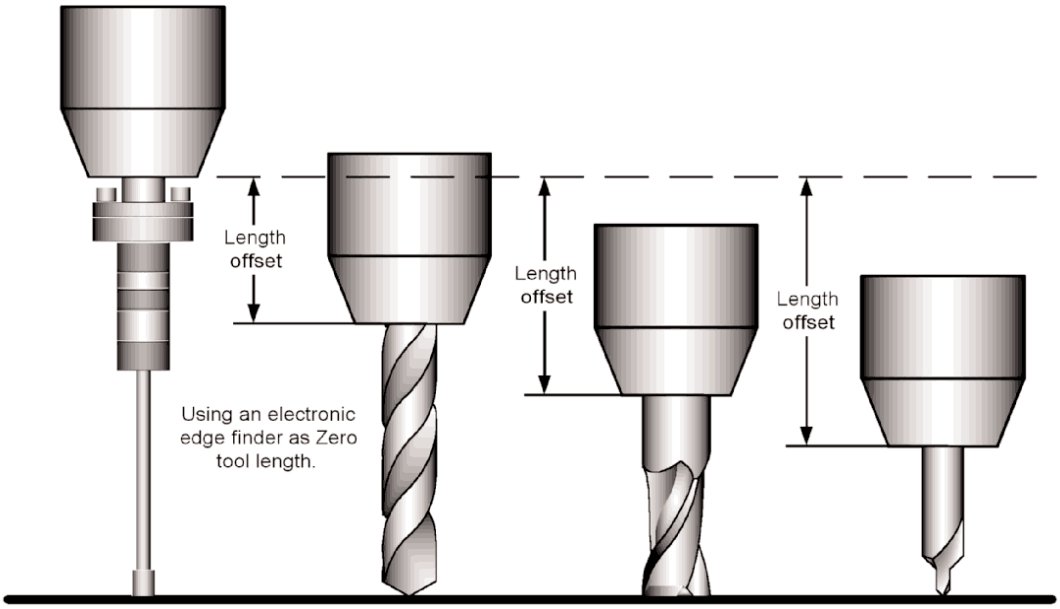
- From Setup, highlight "Tool Library" then press **ENTER**.
- Highlight the tool you want to edit, then press **ENTER**.
- Make the necessary changes, then press **USE**.
- Continue with Setup or press the **USE NEW SETTINGS** softkey.

Using the Tool Library with Repeatable Tool Length Offsets

If you have repeatable tooling, keeping a tool library for your frequently used tools, saves you time by not requiring you to enter the tool information every time you program the tool.

When running G-code programming, the tool length for each tool in the program is retrieved from the library. For example, in a G-code program, "T1 M6" will cause **MILLPWR** to prompt the operator to load tool 1 into the spindle. The tool length offset from tool #1 in the **MILLPWR** tool library is retrieved and used to adjust the Z-axis position. The code T2 uses the length offset of tool 2, etc. This feature takes advantage of your repeatable tooling systems, freeing the operator from having to reset the Z-datum for each tool change.

If your Tooling is not repeatable, you should not place any length information into the tool library.



All tool lengths originate from datum. There are several different ways to establish datum, such as “touching off” with an empty spindle, using a repeatable standard or using an electronic edge finder (refer to **Establishing Datum**).

If you begin setting the tool length offsets by setting Datum using an electronic edge finder it must have a fixed and repeatable length. The current tool information cannot have a length value. All of the tool length offsets in the program will be the difference in length between the tool and the edge finder.

When defining tool length offsets in the Tool Step, touch each tool to the same surface and press the **TEACH TOOL LENGTH** softkey to establish the tool length.

Once you have identified the tools you will be using, you can program the length offsets into the tool steps.

Entering the first tool

Decide on a method that you will use to set your Z-axis datum. You can choose to set your Z-axis datum with the spindle nose (without a tool inserted), using a repeatable standard, or using a 3-D electronic edge finder. To begin your Tool Library, set the current tool information in the status bar so the tool length is zero. In our example, we begin with the spindle nose to set datum zero. If you wish to use the ACU-RITE 3-D electronic edge finder, or a tool holder with a repeatable standard, then place it into the spindle instead.

- Remove any tool from the spindle.
- From the DRO screen, press the **TOOL** key.
- Enter zero for the diameter and zero for the tool length.
- Press **GO**.
- An Operator intervention message will appear asking you to "use" the entered tool.
- Press **GO** again.
- **MILL*PWR*** is now set with a zero tool length offset.
- Press the **DATUM** softkey.
- Touch the spindle nose to a solid surface such as the top of a vice or your workpiece.
- Press the **Z=0** softkey, then the **USE** key.

Datum zero for the Z-axis is now set to the surface that the spindle nose touched. Use this same location to "teach" each tool length in the tool library.

- Press the **SETUP** key and highlight "tool library."
- Press **ENTER**.
- Highlight the tool number in the tool library list, for the tool you are setting up and press **ENTER**.
- Use the numeric keypad to enter the tool's diameter.
- Highlight the "unit" field. If the diameter you entered is metric, press the **MM** softkey.
- Highlight the length field.
- Place this tool into the spindle and touch its tip to the same surface you used to set the Z-axis datum Zero.
- Press the **TEACH TOOL LENGTH** softkey. The length offset for this tool will appear in the length field.
- Highlight the "Type" field.
- Press the **TOOL TYPES** softkey.
- Highlight the type of your tool from the list of available tool types and press **ENTER**.
- Press **USE**.
- Repeat this procedure for all additional tools then, press **USE NEW SETTINGS** to complete the setup.

After the tool information is established in the library, you can edit the values as needed to complete the setup.

Using the Tool Library

Whenever you program a tool step, you will see a **TOOL LIBRARY** softkey. Press the **TOOL LIBRARY** softkey and highlight the tool you wish to use. Press **ENTER**, then **USE**, and the tool step is programmed. It is important to understand that the tool information that is retrieved from the library remains in the program.

If you change a tool in the tool library, the tool information in all of your programs will not change.

Your programs will continue to prompt for the tools you originally programmed. If you wish to update a tool step in a program, highlight the tool step you wish to change, press **ENTER**, press the **TOOL LIBRARY** softkey and highlight the tool you wish to use. Press **ENTER**, then **USE**, and the tool step is updated for the new tool.

Setting Datum

Once you have a library of tools established, you can use any one of the known tools to establish Datum. To establish Datum with a tool from the tool library.

- From the DRO view, press the **TOOL** key.
- Press the **TOOL LIBRARY** softkey.
- Highlight the tool you would like to use from the tool library list and press enter.
- Press the **GO** key. An Operator intervention message will appear asking you to “use” the selected tool.
- Place the selected tool in the spindle.
- Press the **GO** key to confirm that you have installed the tool.
- Follow the instructions for setting datum in your operation manual.

Once Z-axis datum is set with a known tool from the tool library, all of the tools in the tool library can be used without the need to reset datum.

It is extremely important to set Z-axis datum on a solid surface with a tool from the library after running a tool step with this tool, or by using the identical method used when setting your first tool, before you input additional tools to the library. If this is not done, your tool length cannot be determined correctly.

Changing between Tools in the Tool Library when in DRO view

To change tools when in the DRO view,

- From the DRO view, press the **TOOL** key.
- Press the **TOOL LIBRARY** softkey.
- Highlight the tool you would like to use from the tool library list and press enter.
- Press the **GO** key.
- An Operator intervention message will appear asking you to "use" the selected tool.
- Place the selected tool in the spindle.
- Press the **GO** key to confirm that you have installed the tool.

Changing to a Tool of unknown length when in DRO view

- From the DRO view, press the **TOOL** key.
- Enter the tool diameter and if you wish, the type.
- Press the **GO** key.
- An Operator intervention message will appear asking you to "use" the selected tool.
- Place the selected tool in the spindle.
- Press the **GO** key to confirm that you have installed the tool.
- Set the Z-axis datum following the instructions found in your operation manual.

At this point, your tool library length offsets are not in effect. Re-establishing the Z-axis datum with a tool from the tool library will reestablish the tool length offsets.

- From the DRO view, press the **TOOL** key.
- Press the **TOOL LIBRARY** softkey.
- Highlight the tool you would like to use from the tool library list and press enter.
- Press the **GO** key.
- An Operator intervention message will appear asking you to "use" the selected tool.
- Place the selected tool into the spindle.
- Press the **GO** key to confirm that you have installed the tool.
- Reset the Z-axis datum following the instructions found in your manual.

Using your tool library within a program

- When programming, press the **TOOL** key.
- Press the **TOOL LIBRARY** softkey.
- Highlight the tool you would like to use from the tool library list.
- Press the **ENTER** key, then the **USE** key.

When the program is executed and the tool step is reached, an Operator intervention message will appear asking you to "use" the programmed tool.

- Place the selected tool in the spindle. A **MOVE TABLE** softkey is available to help you move away from the workpiece if necessary. You can program a position move prior to the tool step to automatically move off the part before changing the tool.

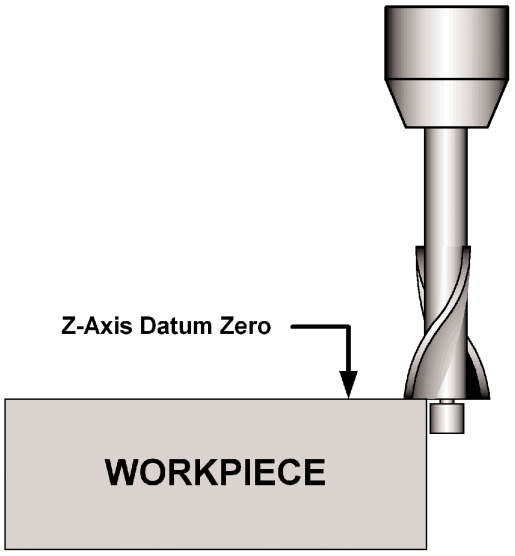
- Press the **GO** key to confirm that you have installed the tool. The program will resume.

Changing to a Tool of unknown, but repeatable length when executing a program

When running a program, it is easy to use tools of unknown but repeatable lengths, such as a counter bore placed in an end mill holder. As long as the counter bore does not leave the end mill holder, its length will repeat each time it is placed in the spindle. Since the tool length remains the same, you can teach this length to **MILL*PWR*** when creating the program without having to enter the tool into the Tool Library.

- Be sure the Z-axis datum was set using a tool from the library (See: Using Your Tool Library above).
- When programming the tool step, enter the tool diameter, then place the tool in the spindle and touch Z-datum.
- Highlight the length field and press the **TEACH TOOL LENGTH** softkey. The tool length offset will appear in the length field.
- Enter any of the remaining information you need in the tool step and press **USE**.

When this tool step is executed, the length offset will take effect and the tool tip will be offset correctly.



Changing to a Tool of unknown, non-repeatable length when executing a program

When running a program, it is possible to switch between tools of known length from the library, to tools that have a non-repeatable length such as a counter-bore held in an R-8 collet. Follow this program as an example:

Step 1 is a tool step, tool 1-a center drill-programmed from the tool library and is repeatable.
Step 2 in the program is a bolthole pattern.
Step 3 is a tool step using tool 2-a drill-from the library and is also repeatable.
Step 4 is the same hole pattern repeated.
Step 5 is a tool step programmed as a counter bore of unknown length and does not repeat.
Step 6 is the same hole pattern repeated.

For this example, two identical parts are being machined. The first part is in place and datum has been set using tool 1 from the library as the current tool. The top of the workpiece is Z-zero. When the **GO** button is pressed, **MILL*PWR*** will see that step one is asking for the same tool as the current tool and will proceed to the next step (Step 2) and drill the holes. When the next tool step (Step 3) is executed, you will be prompted to change tools to the drill. Simply change the tool and press **GO** to continue. The programmed tool offset will compensate for the difference in tool length as it drills the holes (Step 4). The next tool (Step 5) is of unknown length. When this step is executed, and you are prompted to change tools. With this tool change:

- Press the **DATUM** softkey. The DRO screen will appear with the set datum window.
- Place the tool (counter bore) in the spindle and touch the cutting edge to Z-axis Datum Zero (the top of the workpiece in this example).
- Press the **Z=0** softkey and then **USE**. The tip of the tool is now set at datum zero.
- Press GO to continue with the program.

Now that the Z-axis datum was set using a tool of unknown length, tools one and two will not repeat. The Z-axis datum must be reset once again using a tool from the library. This can be done when step 1 is run on the second part. After the part has been changed:

- Press GO. You will be prompted to change tools to the center drill (Step 1).
- Press the **DATUM** softkey. The DRO screen will appear with the set datum window.
- Place the center drill in the spindle and touch the tip to Z-axis Datum Zero (the top of the workpiece in this example).
- Press the **Z=0** softkey and then **USE**. The tip of the tool is now set at datum zero.
- Press GO to continue with the program.

Datum has now been reset with a repeatable tool with a programmed length offset as the current tool, all of the tools with programmed length offsets again relate to datum. The program can continue as usual.

Note: The only information that is mandatory when entering a tool into the tool library, is the diameter. All other information is optional. It is possible to establish a tool library without tool lengths. Z-axis datum will need to be set during each tool step execution of the program.

Scale Factor

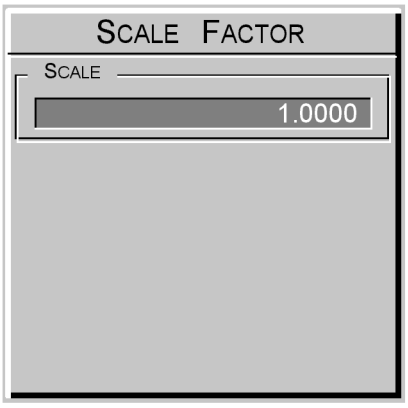
Scale factor is simply a multiplier that lets you expand or shrink your print's dimensions without performing complex calculations or affecting the proportion of your part. Simply enter the number that you'd like your dimensions to be multiplied by, then run your program as you normally would. **MILL*PWR*** will adjust the programmed dimensions (but not the tool size) automatically.

MILL*PWR*'s scale default setting is 1.0000. A value greater than 1.0000 will increase your part's dimensions; a value less than 1.0000 will reduce them. This is saved to the configuration block and will be remembered until it is reset by the user.

MILL*PWR* will display your current scale factor in the status bar (located towards the top of your display screen).

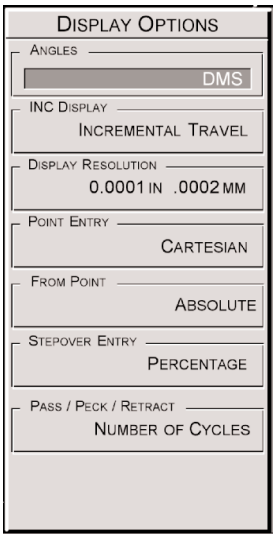
To change the scale factor

- From Setup, highlight "Scale Factor," then press the **ENTER** key.
- Make the necessary changes, then press the **USE** key.
- Continue with Setup or press the **USE NEW SETTINGS** softkey.



Display Options

From the “Setup” menu, select “Display Options” to customize your default settings for display resolution, point entry and similar items.



ANGLES allows you to define how you would like to enter angle values—as decimal degrees, radians or degrees–minutes–seconds (DMS). You should select the angle default setting that satisfies your need most often. In other words, if your angles are usually dimensioned in decimal degrees, you should make **DECIMAL DEGREES** your default setting.

INC DISPLAY enables you to define how you want your incremental values to be displayed. You may select either “Incremental Travel” or “Distance to Go.”

- INCREMENTAL TRAVEL- In this view, the dimensions you program will end at their programmed distance. If you program a 1/4” move, the incremental display will read 0.2500” when you reach the end of your move.

- DISTANCE TO GO- In this view, the dimensions you program will be "preset" into the readout display. Every move will end at zero. If you program a 1/4” move, the incremental display will read 0.0000” when you reach the end of your move.

0.0001 IN 0.002 MM	0.0002 IN 0.005 MM	0.0005 IN 0.010 MM	0.0010 IN 0.020 MM	USE NEW SETTINGS	CANCEL CHANGES

DISPLAY RESOLUTION lets you select the resolution you want to display in the DRO.

POINT ENTRY lets you decide what your default setting will be for entering dimensions for both “From” and “To” points—as either Cartesian (X and Y) or Polar (radius and diameter) coordinates. The method that you do not select will be available as a softkey when you’re defining “From” and “To” points.

FROM POINT allows you to determine whether your "From" point will appear as an absolute value or an incremental value. Absolute values are based upon the distance from datum. Incremental values are based upon the distance from one feature or step in a program to another. No matter which you choose as your default setting, the other will always be available by pressing the **ABS/INCR** key.

Absolute "From" points will appear like this:

```
MILL LINE              002
From
X1      4.0050  ABS
Y1      2.5025  ABS
```

Incremental "From" points will appear like this:

```
MILL LINE              003
From
X1      0.0000  INC    002
Y1      0.0000  INC    002
```

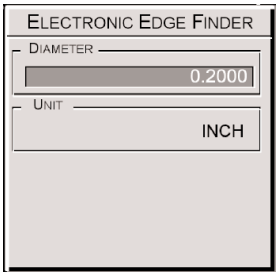
STEPOVER ENTRY enables you to define how much material your tool will remove as it cuts a pocket, slot or ring. To enter a value as a decimal, press the **DECIMAL** softkey. To enter a value using whole numbers, press the **PERCENTAGE** softkey.

PASS/PECK/RETRACT establishes the units that are applied to the "Pass," "Peck" and "Tool Retract" features during programming.

"Pass" refers to the number of cuts that are used to machine an area to its "End" depth.
"Peck" lets you break chips and reduce chip buildup during drilling operations.
"Tool Retract" allows you to program **MILL*PWR*** to raise the tool at regular intervals.

Using the softkeys, select either **NUMBER OF CYCLES** or **DISTANCE**. Your selection will be applied to all three features.

Electronic Edge Finder



An electronic edge finder enables you to "teach" positions, find the center point of a circle, skew a part or locate datum (also known as workpiece zero) by simply "touching off" on the part. The greatest advantage of an electronic edge finder is that it instantly senses when you've made contact —even if you've over-traveled.

MILL*PWR* lets you define the "Diameter" and "Unit" of measure (either inches or millimeters) for an electronic edge finder. Once this information has been entered into **MILL*PWR***, **MILL*PWR*** will automatically compensate for the radius of the tip of the electronic edge finder when performing any of the operations mentioned above.

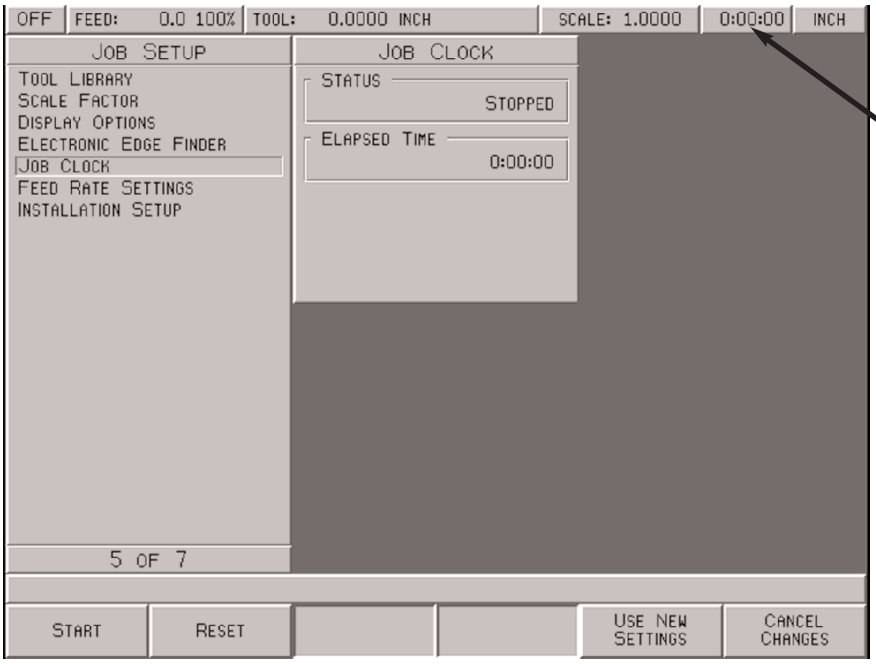
To define the diameter and unit of measure for an electronic edge finder:

- From Setup, highlight "Electronic Edge Finder" then press the **ENTER** key.
- Enter a value for the electronic edge finder's tip diameter and unit of measure (either inches or millimeters) then press the **USE** key.
- Continue with Setup or press the **USE NEW SETTINGS** softkey.

Job Clock

MILL*PWR* includes a job clock that can be used to keep track of the time you spend on each job. The job clock works like a stopwatch, counting in one second intervals after you've pressed the **START** softkey and stopping after you've pressed the **STOP** softkey.

The job clock appears in the "Status" bar and is visible from both the PGM and DRO screens. It reflects how much time has elapsed since the clock was started. Every time you stop the job clock, the time will be added to the "Elapsed Time" field on the "Job Clock" menu and the value in the "Status" bar will be reset to zero. You can use the job clock to time each step of a project and then refer to the "Elapsed Time" for the total time spent on the job.



Job Clock

Note: The maximum amount of time that the job clock will record is 99 hours, 59 minutes and 59 seconds. The value will then roll over to zero and continue counting.

To start the job clock:

- From Setup, highlight “Job Clock” then press the **ENTER** key.
- Press the **START** softkey, and the job clock will begin.

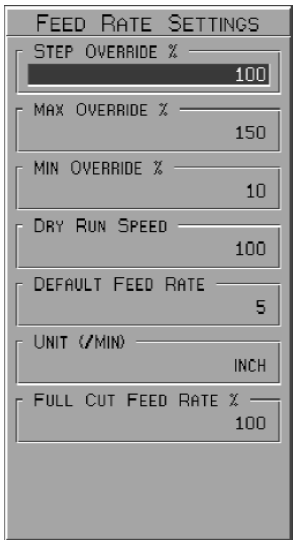
To stop the job clock:

- From Setup, highlight “Job Clock” then press the **ENTER** key.
- Press the **STOP** softkey, and the job clock will stop. The job clock’s current time will be added to the value in the“Elapsed Time” field.
- To restart the job clock, press the **START** softkey.

To reset the job clock and the “Elapsed Time” field:

- From Setup, highlight “Job Clock” then press the **ENTER** key.
- Press the **RESET** softkey, and the job clock and “Elapsed Time” field will be cleared.

Feed Rate Settings



Here you can change the default settings for various speeds and feeds:

STEP OVERRIDE % determines the percentage of change that occurs when you press the **FEED+** and **FEED-** keys.

MAX % establishes the maximum feed rate percentage that the **FEED+** key will allow.

MIN % establishes the lowest feed rate percentage that the **FEED-** key will allow.

DRY RUN SPEED is the speed that the table will move during a dry run.

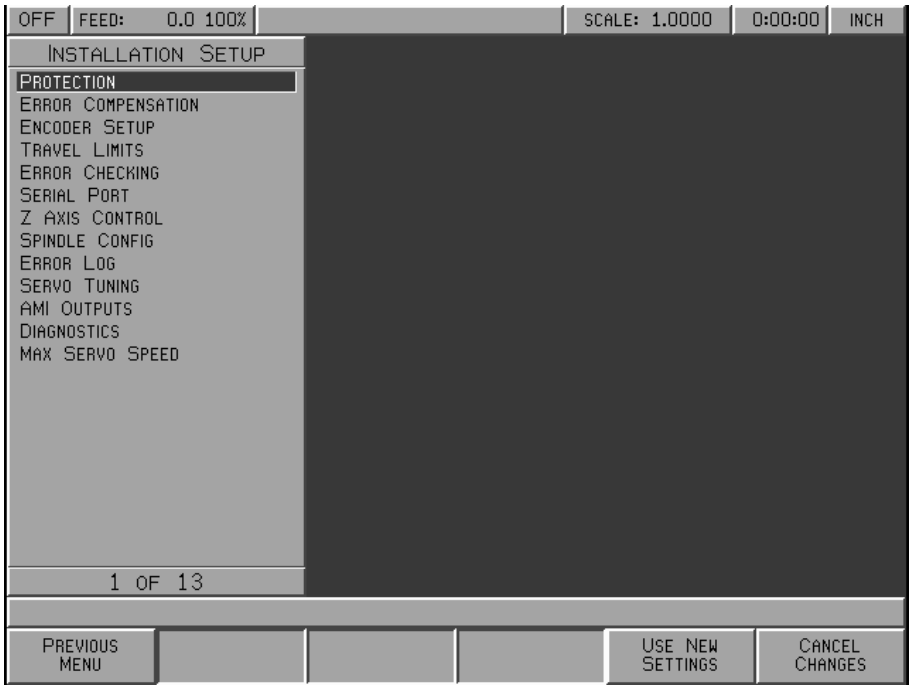
DEFAULT FEED RATE enables you to define the table's and Z-axis' default feed rate for all milling functions—this way, if you forget to enter a feed rate, **MILL*PWR*** will automatically apply your default feed rate to the milling function. We suggest that you set your default feed rate to the rate that you use most often.

UNIT/MIN lets you define your feed rate in either inches per minute or millimeters per minute.

FULL CUT FEED RATE % is the percentage of the programmed feed rate used when the tool is making a full cut (e.g., the beginning cut in a pocket).

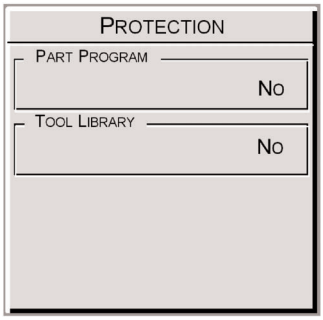
Installation Setup

After you've selected "Installation Setup" from the "Job Setup" list, you will be asked for a passcode. The passcode is located on the page before the table of contents. You can remove it and store it in a separate location to prevent accidental or unauthorized adjustments to your system. Once you've entered the passcode, the "Installation Setup" screen will appear.



Protection

Protection enables you to prevent unauthorized access to your part programs and tool library. With "Part Program" and "Tool Library" set to "Yes," your part programs and tool library cannot be altered—without first changing your "Part Program" and/or "Tool Library"setting's to "No."



To change your protection settings:

- From Installation Setup, highlight **PROTECTION** then press **ENTER**.
- Use the **NO** and **YES** softkeys to disable or enable protection for both the "Part Program" and "Tool Library."
- Press the **USE** key.
- Continue with Setup or press the **USE NEW SETTINGS** softkey.

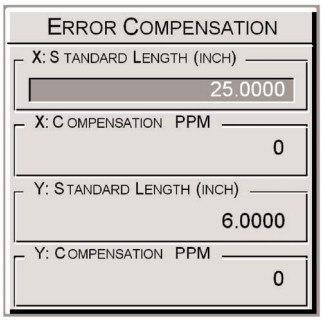
Error Compensation

Before performing linear error compensation, you need to ensure that your machine is properly adjusted. Refer to the manufacturers' instructions for proper adjustment procedures.

MILL*PWR* is able to automatically calculate and compensate for machine tool wear.

If you know the error compensation value in parts per million (PPM), you can enter it directly. Otherwise, use the following procedure to determine the value for each axis:

Note: You can only perform error compensation for one axis at a time.



- Set up a standard of known length (e.g., a gage block), by indicating it in along the axis to be measured. Check that the standard is straight and parallel with the axis you're working with.
- Enter the length of the standard.

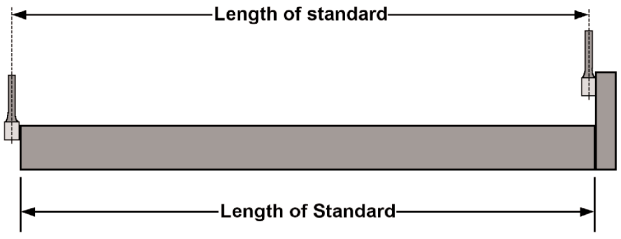
  If you need to touch each end of the standard using opposite edges of the electronic edge finder or mechanical indicator, be sure to include its diameter when entering the length of the standard. (Refer to the graphic below.)

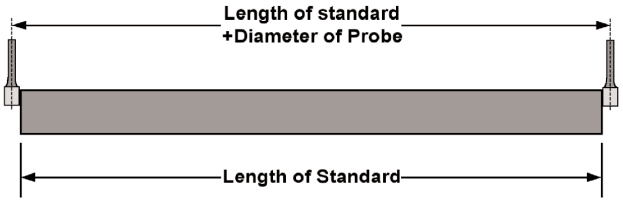
Check between the first and second points several times—note the error value to ensure repeatability.

- Touch one end of the standard and press the **FIRST POINT** softkey.

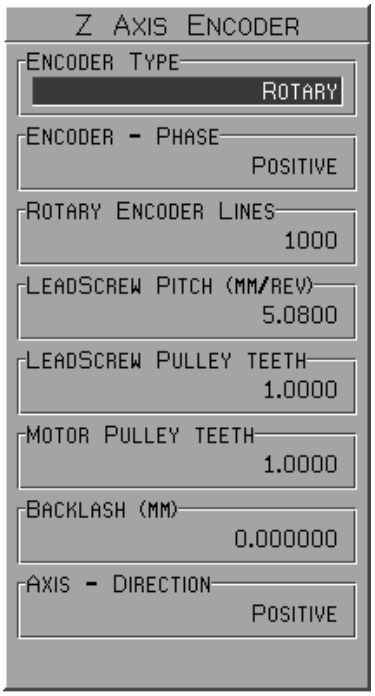
  If you're using an electronic edge finder, you won't have to press the first point softkey.
- Touch the opposite end of the standard and press the **SECOND POINT** softkey. If you're using an electronic edge finder, you won't have to press the second point softkey.

  MILL*PWR* will determine and automatically enter the correct compensation factor in parts per million.
- Continue with Setup or press the **USE NEW SETTINGS** softkey.





Encoder Setup



When your **MILL*PWR*** system was installed, the encoder type should have been established. To change the encoder type, select the appropriate softkey.

Also, when your **MILL*PWR*** system was installed, the encoder count direction should have been established for each axis. Each scale should have been assigned to move in either a positive or negative count direction, depending on how and where the scales were mounted to your machine.

> ⚠ You shouldn't have to change your X- and Y-axes encoder directions. Should you want your count directions changed, please contact your ACU-RITE distributor.

You *can* change the count direction for the Z- and W-axes. The Z-axis measures the movement along the quill, and the W-axis measures movement along the knee.

With **MILL*PWR's*** coupling feature, you can add, subtract, or average the movements of two parallel axes—in this case the Z- and W-axes. Coupling the Z-axis with the W-axis allows you to maintain your Z-axis datum and tool offsets whether you move the Z-axis or the W-axis. When the axes are coupled, both change the Z-axis DRO count value.

Be careful about changing the count direction and then running an old program—you could get some pretty strange results!

To change the encoder direction and coupling settings:

- From Installation Setup, highlight "Encoder Setup" and press **ENTER**.
- Highlight the axis you wish to change and press **ENTER.** Now use the **NEGATIVE** and **POSITIVE** softkeys to change the count direction.
- Highlight "Coupling," and use the **OFF**, **COUPLE Z+W** and **COUPLE Z-W** softkeys to change coupling.
- Press the **USE** key.
- Continue with Setup or press the **USE NEW SETTINGS** softkey.

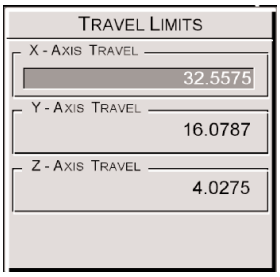
To change the direction of the encoder phase:

- From within the Encoder Setup menu, arrow down and highlight the "Encoder - Phase" field.
- Now use the **NEGATIVE** and **POSITIVE** softkeys to change the phase direction.
- Press the **USE** key.
- Continue with Setup or press the **USE NEW SETTINGS** softkey.

Travel Limits

Travel limits define how far you will allow the table to move along the X- and Y-axes and how far up and down the quill can be positioned along the Z-axis.

Note: You must find home before you can establish your travel limits.



To set your travel limits:

- From Installation Setup, highlight "Travel Limits" and press **ENTER**. The “X-axis Travel” field should be highlighted.
- Move the table to the position where you want your right table travel limit set.
- Press the **SET RIGHT LIMIT** softkey.
- Move the table to the position where you want your left table travel limit set.
- Press the **SET LEFT LIMIT** softkey.

  Note: The X-axis travel field will update to show you the maximum allowable traveling distance between the left and right table travel limits.

- Now highlight the "Y-axis Travel" field.
- Move the table away from you until you've reached the position where you want the inside table travel limit set.
- Press the **SET INSIDE LIMIT** softkey.
- Move the table towards you until you've reached the position where you want the outside table travel limit set.
- Press the **SET OUTSIDE LIMIT** softkey.

  Note: The Y-axis travel field will update to show you the maximum allowable travel distance between the inside and outside table travel limits.

Use the handles or the MOVE TABLE feature to position your table.

If you use MOVE TABLE, a caution message will inform you that the table travel limits are not activated.

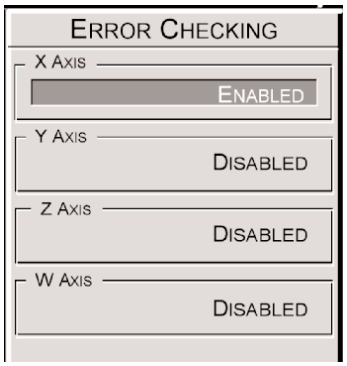
- Highlight the "Z-axis Travel" field.
- Move the quill up until you have reached the position where you want the top travel limit set. Make sure FTO position is located.
- Press the **SET TOP LIMIT** softkey.
- Move the quill down until you have reached the position where you want the bottom travel limit set. Make sure FTO position is located.
- Press the **SET BOTTOM LIMIT** softkey.
- Press the **USE** key.
- Continue with Setup or press the **USE NEW SETTINGS** softkey.

You may change your table travel limits at any time.

Error Checking

With the error checking feature enabled, **MILL*PWR*** can detect scale miscounts and record them in the Error Log. **MILL*PWR*** will stop the program if a scale miscount is detected.

To enable/disable the error checking feature:



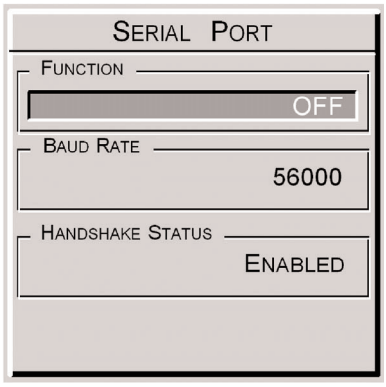
- From Installation Setup, highlight "Error Checking" and press **ENTER**.
- Press either the **ENABLE** or **DISABLE** softkey.
- Press the **USE** key.
- Continue with Setup or press the **USE NEW SETTINGS** softkey.

Serial Port

You can connect **MILL*PWR*** to your PC via the serial port (located on the rear of the operator console). The serial port allows you to store part programs on your PC.

To activate the serial port:

- From Installation Setup, highlight "Serial Port" and press **ENTER**.
- Press the **REMOTE STORAGE** softkey to change the "Function" setting from "Off" to "Remote Storage."
- Highlight "Baud Rate."
- Define the Baud rate by pressing the **LOWER** or **HIGHER** softkeys. The Baud rate you select must match your PC's Baud rate.



- Highlight "Handshaking."(The handshaking feature verifies that **MILL*PWR*** and your PC are communicating properly during file transfers.)
- Press the **ENABLED** softkey.
- Press the **USE** key.
- Continue with Setup or press the **USE NEW SETTINGS** softkey.

Z-axis Control

Your **MILL*PWR*** is shipped with Z-axis control enabled. However, you can easily disable the Z-axis control at your convenience—for example, when you want to run a program that was created using a two-axes system.

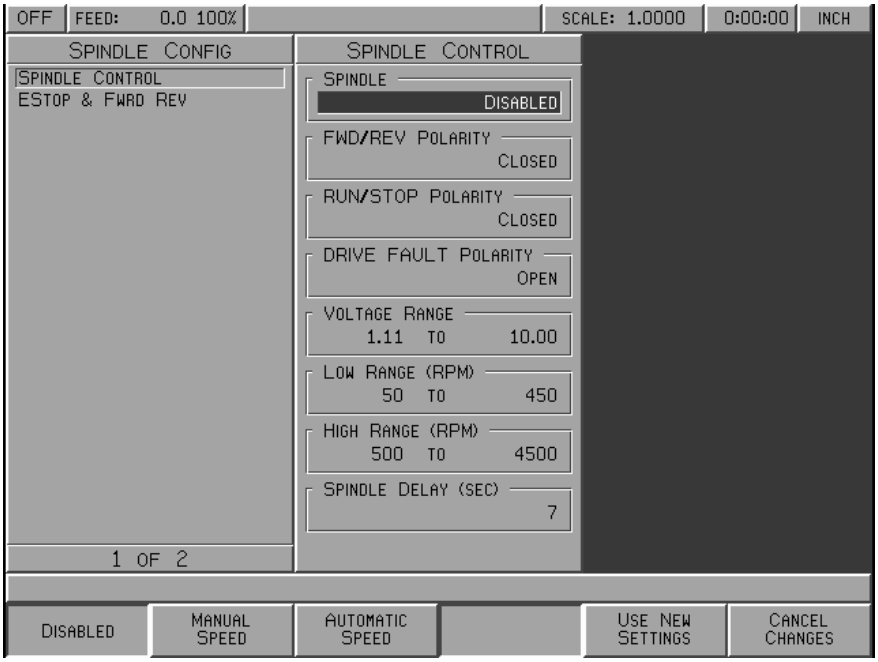
- To operate **MILL*PWR*** as a three-axes system (with X, Y and Z motion control), select the **ENABLE** softkey.
- To use **MILL*PWR*** as a two-axes system (with only X and Y motion control), select the **DISABLE** softkey.

Spindle Control

If you're using a milling machine that has an electronic variable-speed spindle, **MILL*PWR*** may be able to control the spindle for you through the Auxiliary Machine Interface option.

With the spindle control enabled, **MILL*PWR*** will apply the direction and speed parameters you enter into the "Set Tool" step of a program. Depending on your spindle's motor, you may be able to control the spindle's speed and direction, or just the direction.

If the spindle control is *disabled*, the settings you program in "Set Tool" will be displayed in an Operator Intervention Message (OIM) when you run the program. You'll be prompted to set the speed and direction manually.



Error Log

MILL*PWR* includes a built-in error log that will record system assertion failures. If an assertion failure is detected, **MILL*PWR*** will automatically record the date, along with a brief description of the failure.

Messages can be cleared by pressing the **CLEAR** key, then pressing **YES** to proceed or **NO** to cancel. It's best to let messages accumulate to establish a history.

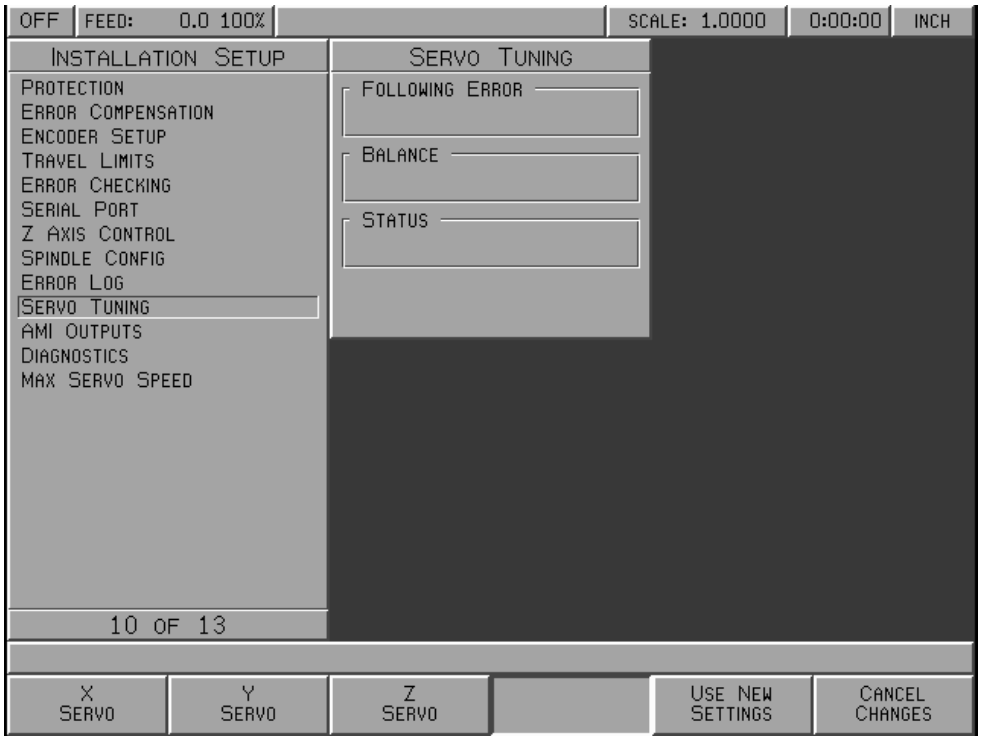
To save these records for future reference, place a 3 ½" floppy disk into the disk drive and press the **SAVE FLOPPY** softkey.

Servo Tuning

The servo motors for each **MILL*PWR*** system are properly tuned upon installation. However, you can fine tune your system periodically using the Auto Tuning feature. Before beginning the Auto Tuning procedure, first make sure your **MILL*PWR*** system software is version 3.2.8 or higher.

The Auto Tuning method is done completely through the **MILL*PWR*** console. Therefore, there is no need to open the servo motors. Since all adjustments are made from the console, the changes will not affect the original amplifier setting values.

When using the Auto Tuning feature, you should tune each axis (X-, Y- and Z) individually—to achieve optimal performance. Both the Balance and the Gain for the axis will need to be adjusted.



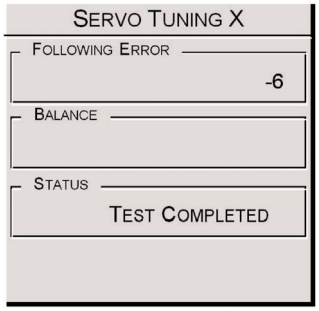
If the servo motors cannot be tuned automatically, the following message will appear: "Not enough adjustment, please use manual mode."

Please contact your ACU-RITE distributor.

Automatically Tuning the Servo in Console Mode

1. Make sure the axis you are tuning is roughly in the middle of its travel (They will be moving approximately two (2) inches in each direction.)

   Note to Installer: No RS-232 cable should not be connected to any axis. Setting value tables could become contaminated.

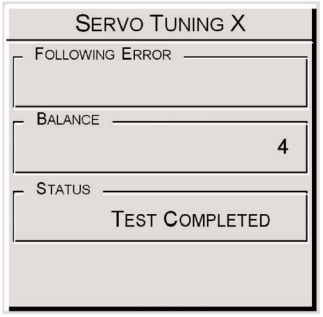


2. Power up the **MILL*PWR*** console, make sure the E-Stop button is out, and home the axes as you would normally.

3. Press the "Setup" key.

4. Arrow down to highlight "Installation Setup" and press **ENTER**.

5. When prompted type in the required passcode and press **ENTER**.

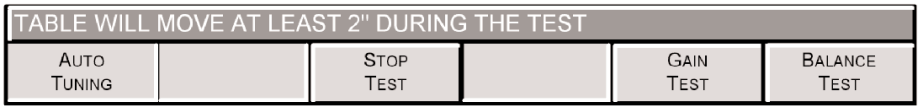


6. Arrow down to "Servo Tuning" and press **ENTER**.

7. Select the axis you want to tune by pressing the appropriate soft key.

8. The prompt bar should indicate you are in Console Mode.

9. Press the **AUTO TUNING** soft key, then press the **BALANCE TEST** soft key.



10. The console will begin adjusting the balance values until it finds the value that keeps the axis as motionless as possible.

> ⚠ **Do not touch the table during the balance test.**

11. When the Auto Tuning process is completed, the prompt bar will read "Test Completed." Now proceed with the Gain Test.

12. Make sure the axis you are tuning is roughly in the middle of its travel. Press the **GAIN TEST** soft key.

13. The control will cause the axis to begin moving back and forth approximately two (2) inches in each direction.

14. While the axis is moving, the control automatically adjusts the Gain value until the “Following Error” is at its lowest value.

15. Once the control has reach the optimal Gain value, it will automatically stop axis motion and indicate it is done by displaying “Test Completed.”

16. The correct values for the Gain and Balance tests have now been established. Now you will need to exit and save the new values in to the console.

17. Press the **STOP TEST** soft key, then press the **USE NEW SETTINGS** soft key.

18. Power down the console and restart to begin normal operation and have the updated values implemented.

AMI Outputs

In Position Relay

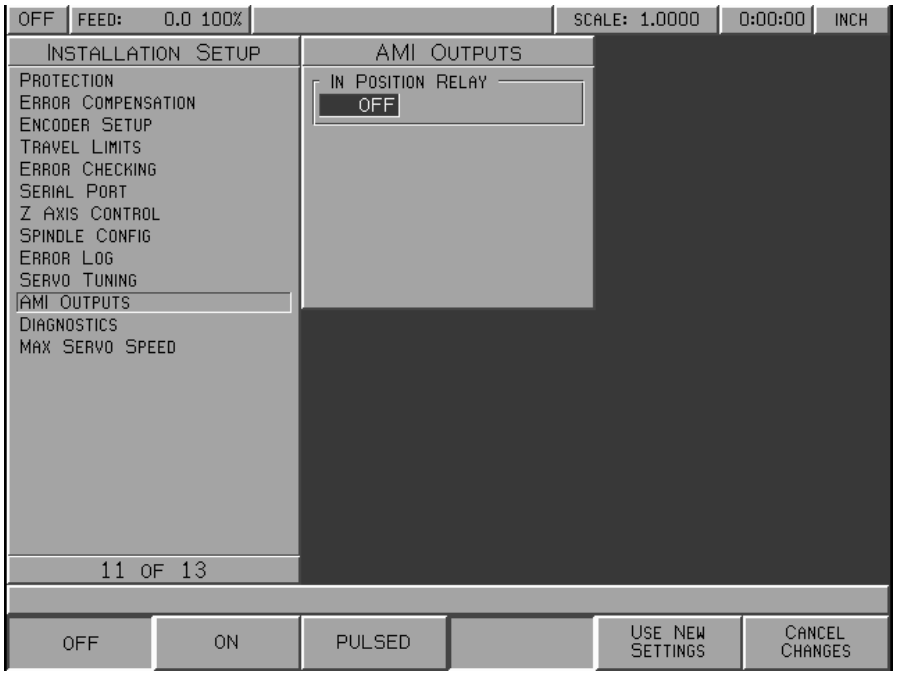
The Auxiliary Machine Interface (AMI) has the ability to send a signal to external devices. This signal is sent when **MILL*PWR*** is in position for Position Drill steps and Hole pattern steps. This is when the table has moved to either the X, Y coordinates of the position drill step or to each hole of a hole pattern.

Choose the appropriate signal for your application from the three options explained below.

ON: When the X, Y coordinates for the position drill step or when each hole of a hole pattern is reached, a signal is sent for output. This signal will continue to be output until the **GO** button is pressed, a GO signal is received through the AMI, the table stop is activated, the program is stopped, or the program ends.

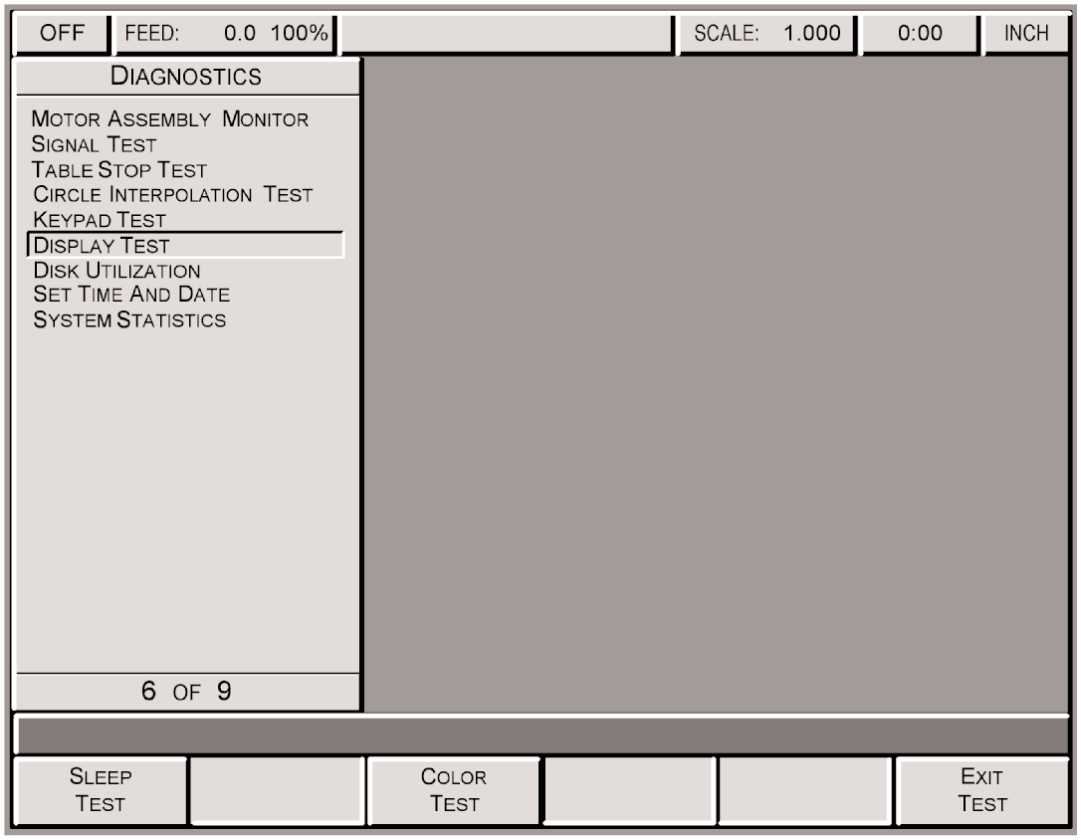
OFF: No signal is sent.

Pulsed: With this option the signals will only be sent for the timed intervals that you establish. Enter a time interval between 0.1 and 999.9 seconds. When the X, Y coordinates for the position drill step, or when each hole of a hole pattern is reached, a pulsed signal will be sent. This signal will continue until either the pulse time interval expires, the **GO** button is pressed, a GO signal is received through the AMI, the table stop button is activated, an E-Stop input is detected, the program is stopped or the program ends.



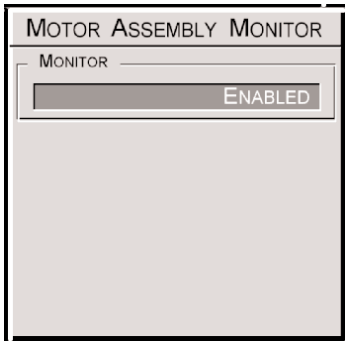
Diagnostics

The diagnostics section houses information about your system's mechanical operations, memory availability and machine usage. Tests are also available to help you monitor your system's overall performance.



Motor Assembly Monitor

When the motor assembly monitor is enabled, it shows the real-time operating status of each axis on the DRO screen.



X	POWER ON	Y	POWER ON	Z	POWER ON
INC	0.0000	INC	0.0000	INC	0.0000
ABS	0.0000	ABS	0.0000	ABS	0.0000
STATUS	OFF	STATUS	OFF	STATUS	OFF
TEMP	15°C	TEMP	14°C	TEMP	15°C
CURRENT	-24	CURRENT	17	CURRENT	11

Top Bar (Power Status) Reports on the AC power status (On or Off); also reports system failures and emergency stop conditions when they occur.

INC Displays the table's incremental position (its position relative to incremental zero).

ABS Displays the table's absolute position (its position relative to datum).

Status Reports on each motor's operating status (On, Off, Not Present or Fault).

Temp Shows the internal air temperature for each motor. Temperatures should not exceed 75° C.

Current Monitors the motor's current voltage. An "X" after the value indicates that the voltage inrush bypass circuit is active. It can also be seen when the E-stop button is pressed and then released. The "X" should disappear within 2 - 3 seconds after the system is powered up.

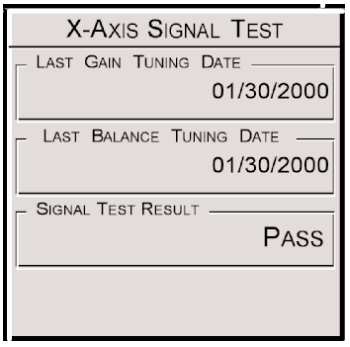
To enable or disable the motor assembly monitor:

- From the Diagnostics screen, highlight "Motor Assembly Monitor" and press **ENTER**.
- Press either the **ENABLED** or **DISABLED** softkey. (If the **ENABLED** softkey is selected, the motor assembly monitor display will appear on the DRO screen.)
- Press the **USE** key.
- Continue with Diagnostics or press the **USE NEW SETTINGS** softkey.

Signal Test

The signal test shows when an axis was last tuned and whether or not the digital-to-analog converter is emitting an analog signal. When the test is complete, either “Pass” or “Fail” will appear in the “Signal Test Result” field. If a failure is reported, check the message bar prompt for details.

To display the signal test:



- From the Diagnostics screen, highlight "Signal Test" and press **ENTER**.
- Press the softkey for the appropriate axis.
- Press the **EXIT TEST** softkey to close the “Signal Test” screen.
- Continue with Diagnostics or press the **USE NEW SETTINGS** softkey.

Table Stop Test

The table stop test shows whether or not an emergency stop (E-stop) condition has been detected or the internal switch has been pressed.

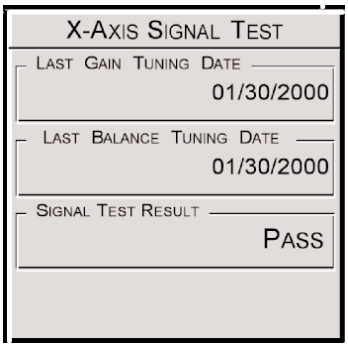
E-stop Signal Status Reports on whether or not the system detects an emergency stop condition. An E-stop condition can originate from the operator console, Z-axis, or optional AMI.

Table Stop Button Displays the status of the table stop button (Pressed or Released) on the operator console.

Z-Limit If Z-axis control is enabled, a “Fault” or “No Fault” message will appear. A fault indicates that at least one of the limit switches on the spindle has been disengaged.

To display the table stop test:

- From the Diagnostics screen, highlight "Table Stop Test" and press **ENTER**. The test will run automatically.
- When the test is complete, press the **EXIT TEST** softkey to close the “Table Stop Test” screen.
- Continue with Diagnostics or press the **USE NEW SETTINGS** softkey.



Circle Interpolation Test

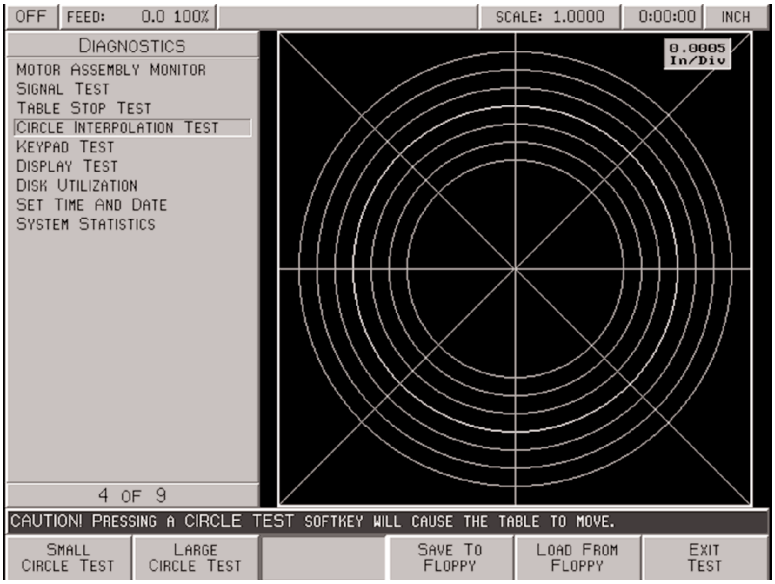
MILL*PWR* includes two circle interpolation tests. The small circle test checks for machine-related problems—an unbalanced ballscrew or poor gib adjustment, for example. The large circle test checks whether or not the motors are in tune.

The circle should maintain its original shape (fluctuations up to 0.0005" are normal). If fluctuations exceed 0.0005", the system may require adjustment (refer to the **Trouble-shooting Guide** for recommended solutions). The examples on the next page illustrate some of the problems that can be detected. For accurate results, do not run the circle test while the spindle is under a cutting load.

You have the option of saving the test results onto a 3 ½" floppy disk for later review.

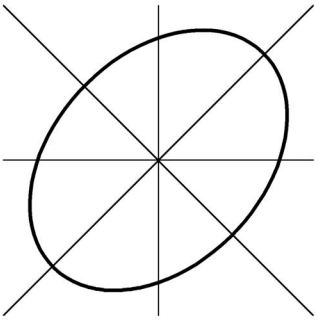
To run the circle interpolation test:

- From the Diagnostics screen, highlight "Circle Interpolation Test" and press **ENTER**.
- Check that the table has at least 3½" of clearance for each axes.
- Press either the **SMALL CIRCLE TEST** or **LARGE CIRCLE TEST** softkey.
- (Optional) When the test is completed, insert a 3 ½" floppy disk into the **MILL*PWR*** disk drive and press the **SAVE TO FLOPPY** softkey. After **MILL*PWR*** has finished saving, remove the disk.
- Press the **EXIT TEST** softkey to close the "Circle Interpolation Test" screen.
- Continue with Diagnostics or press the **USE NEW SETTINGS** softkey.

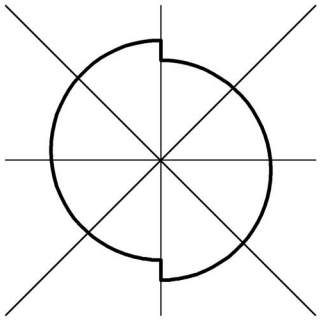


Examples of Circle Interpolation Test Results

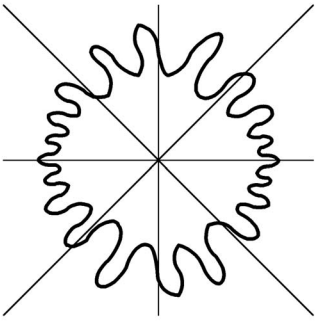
Please note that the graphics are exaggerated for clarity.



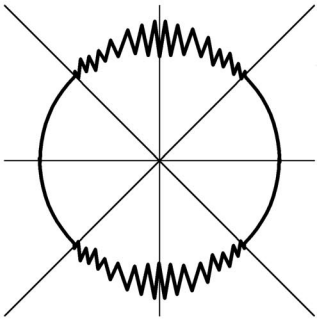
Following Error caused by servo gain misadjustment



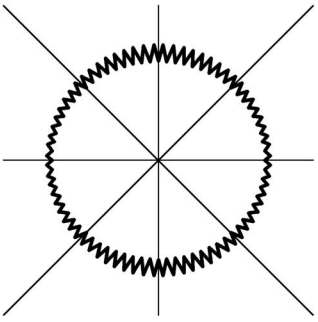
Backlash from lost motion or play in the guideways



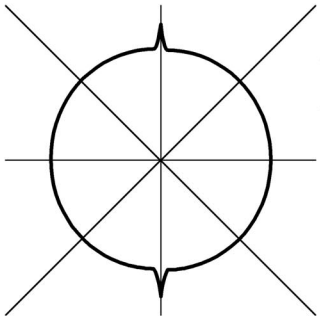
Cyclic Error caused by a flaw in the ballscrew or ballscrew mounting



Stick-Slip of an axis, esp. at low speeds (in this case, Y-axis)



Machine Vibration in an axis (in this case, Y-axis)



Axis Reversal Spike shows an axis sticking when velocity reaches zero (in this case, Y-axis)

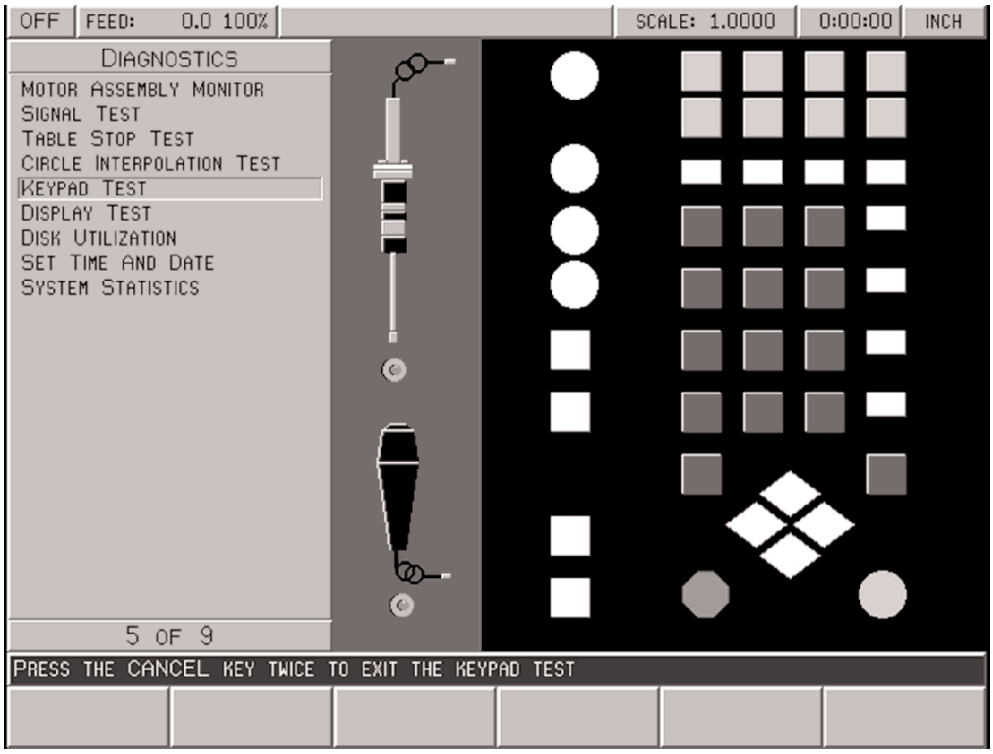
Keypad Test

The keypad test verifies that all of the keys, the remote pendant switch and the optional electronic edge finder are functioning properly.

When you press a key, the corresponding graphic should shrink; when you release the key, the graphic should return to its full size. If the graphics don't change at all, it is likely that the key is not functioning properly. If the graphic shrinks, but does not return to normal when the key is released, the key may be sticking. Bullets (•) mark the keys that have already been tested.

To run the keypad test:

- From the Diagnostics screen, highlight "Keypad Test" and press **ENTER**.
- Press the key(s) on the keypad that you want to test. Press the remote pendant or use the electronic edge finder to test their status.
- Press the **CANCEL** key on the keypad *twice* to exit the "Keypad Test" screen.
- Continue with Diagnostics or press the **USE NEW SETTINGS** softkey.



Display Test

The display test is used to verify that the system's "sleep" mode is working properly and that all of the pixels in the LCD are functioning.

Pressing the "Sleep Test" softkey will activate the sleep mode (the screen will go blank). Press any key or move the table to return the screen to its normal view.

Each time you press the "Color Test" softkey, the display will flash a different color. Keep pressing the softkey until the sequence is complete and the screen returns to the normal view.

To run the display test:

- From the Diagnostics screen, highlight "Display Test" and press **ENTER**.
- Press either the **SLEEP TEST** or **COLOR TEST** softkey.
- If you pressed the **SLEEP TEST** softkey, press any key or move the table to return the screen to its normal view.

  If you selected the **COLOR TEST** softkey, press it repeatedly and scroll through the color sequence (black, white, red, green, blue). The test is complete when the normal view returns.
- Press the **EXIT TEST** softkey to close the "Display Test" screen.
- Continue with Diagnostics or press the **USE NEW SETTINGS** softkey.

Disk Utilization

The disk utilization screen shows **MILL*PWR***'s memory capacity and how much memory is still available. Like a PC, **MILL*PWR*** uses memory each time a file is saved onto the system's internal memory. When a file is deleted, memory is restored.

Total Clusters — Total number of clusters available on the **MILL*PWR*** system. One cluster equals 8,192 bytes of memory.

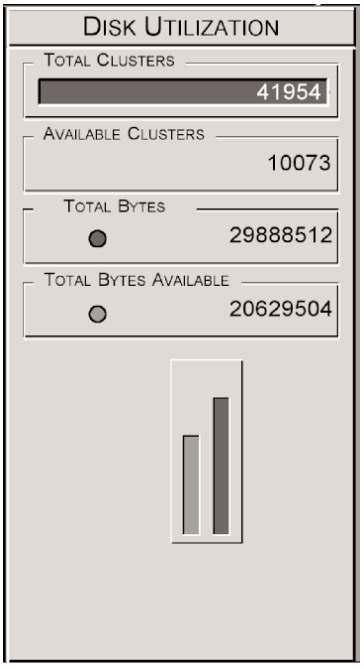
Available Clusters — Number of clusters still available for use.

Total Bytes — Total amount of memory (measured in bytes) shipped with the **MILL*PWR*** system.

Available Bytes — Amount of memory (measured in bytes) still available for use.

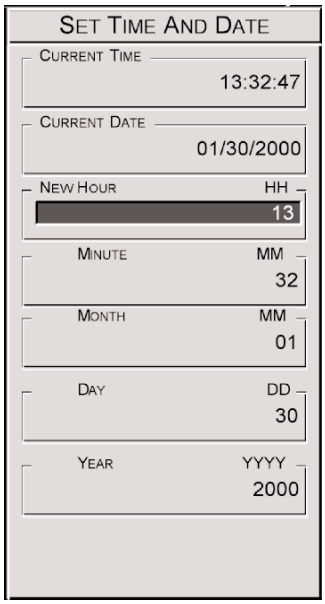
To view the disk utilization screen:

- From the Diagnostics screen, highlight "Disk Utilization" and press **ENTER**.
- Press the **EXIT TEST** softkey to close the "Disk Utilization" screen.
- Continue with Diagnostics or press the **USE NEW SETTINGS** softkey.



Set Time and Date

The time and date appear on the error log and on any saved part program. You can reset the time and date at any time. Enter four digits for the year.



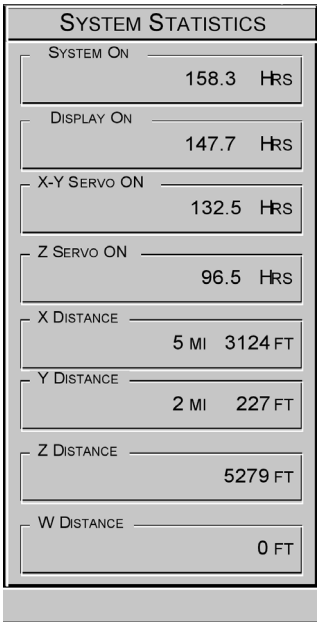
To change a setting:

- From the Diagnostics screen, highlight "Set Time and Date" and press **ENTER**.
- Highlight the field you wish to update.

  Using the keypad, enter the appropriate number.
- Press the **USE NEW SETTINGS** softkey to close the “Set Time and Date” screen.
- Continue with Diagnostics or press the **USE NEW SETTINGS** softkey again.

System Statistics

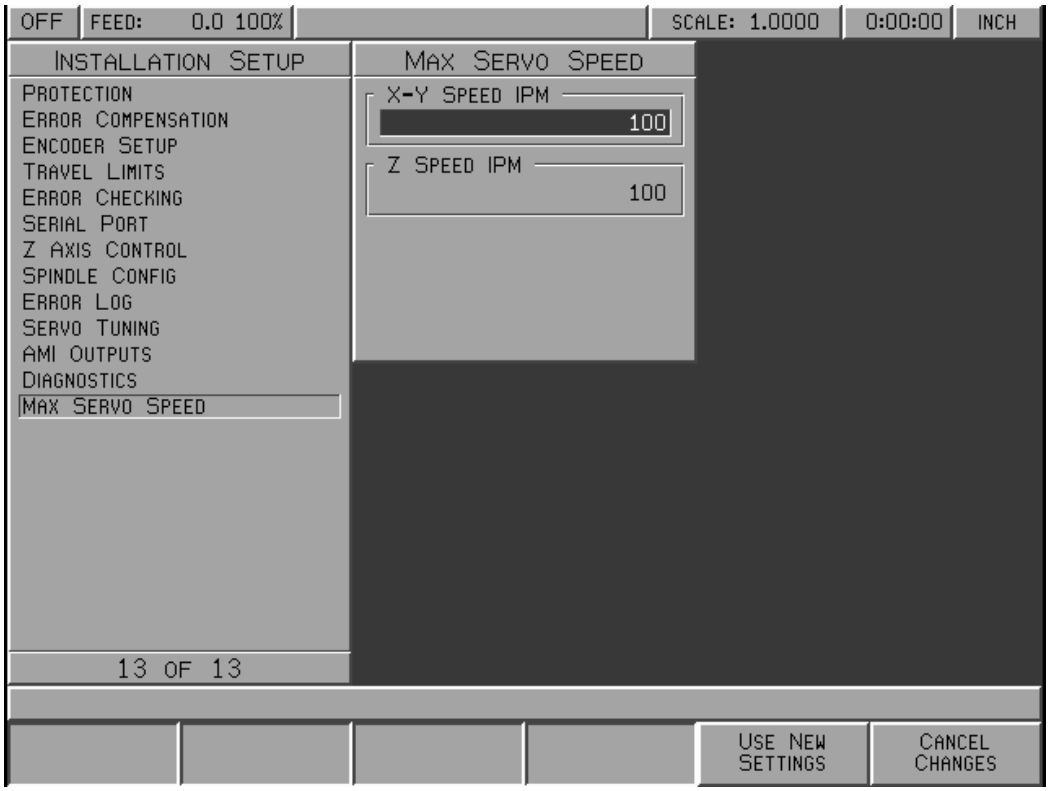
The system statistics keep track of the machine’s usage. The time that the **MILL*PWR*** system, the display and the servo motors have been running is displayed in hours. The “distance” values for X, Y, Z and W reflect how far the scales have traveled for each axis.



To view the system statistics:

- From the Diagnostics screen, highlight "System Statistics" and press **ENTER**.
- Press the **EXIT TEST** softkey to close the “System Statistics” screen.
- Continue with Diagnostics or press the **USE NEW SETTINGS** softkey.

Max Servo Speed



MILL*PWR* can move each axis a maximum of 100 inches per minute (IPM). This parameter allows you to establish a lower maximum speed for the X, Y, axes and/or the Z axis.

To lower the maximum servo speed

- Using the arrow keys, highlight the X-Y SPEED IPM field.
- Using the keypad, enter the new speed for the X, and Y, axes in inches per minute.
- Using the arrow keys, highlight the Z SPEED IPM field.
- Using the keypad, enter the new speed for the Z axis in inches per minute.
- Continue with setup or press the USE NEW SETTINGS softkey.

Remote Storage

When you create programs using **MILL*PWR***, you can save them on your PC using the remote storage feature. The remote storage feature enables you to free up **MILL*PWR***'s internal memory. Plus, your PC is the ideal place to back up important programs and organize them into one convenient location.

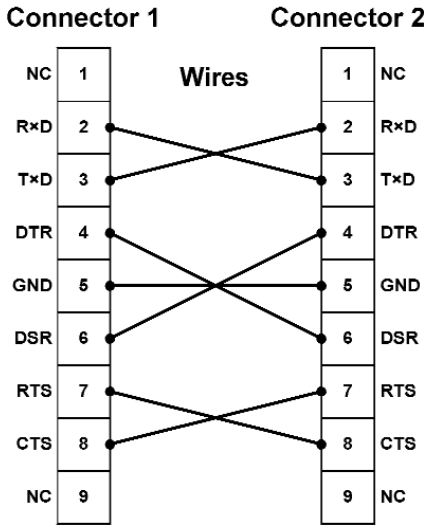
Equipment

To set up remote storage, you will need the following items:

- **MILL*PWR***
- IBM-compatible PC
- Serial cable
- Remote Storage software

Choosing a Serial Cable

Before **MILL*PWR*** and your PC can communicate, they need to be connected to each other with a serial cable. The graphic below illustrates the necessary cable wire configuration. Check that the cable you purchase conforms to this wiring diagram:

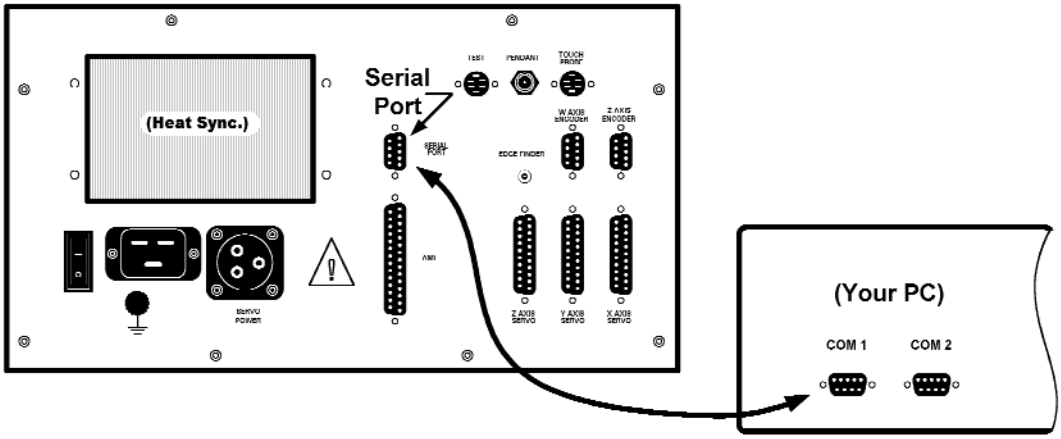


Connecting MILL*PWR* to Your PC

> ⚠ Power down both **MILL*PWR*** and your PC before connecting the serial cable.

Connect one end of the cable into the serial connector located on the rear of your **MILL*PWR*** operator console. Connect the opposite end of the cable into an available COM port on your PC.

Note: The COM port must be *available*—that is, not already assigned a piece of hardware. You cannot simply disconnect a piece of hardware and connect the serial cable. If there's no COM port recognized as being available by your PC, then you will need to make one available (consult your PC owner's manual).



Installing the Remote Storage Program onto Your PC

You can install the Remote Storage program onto your PC using either Microsoft® Windows® 95/98 or MS DOS® operating systems. Choose the system you will use, then follow the applicable steps below.

For Windows 95/98:

1. Power up your PC.
2. Insert the "Remote Storage" disk into your PC's floppy disk drive.
3. Click on "START."
4. Click on "Run," then type **a:install** and click "OK." This copies all of the files from the disk and onto your PC.
5. Instructions on how to use Remote Storage will be displayed on your PC screen.
6. Close the DOS window. A Remote Storage icon should now appear on your PC screen. Double-click on the icon to run the Remote Storage program.

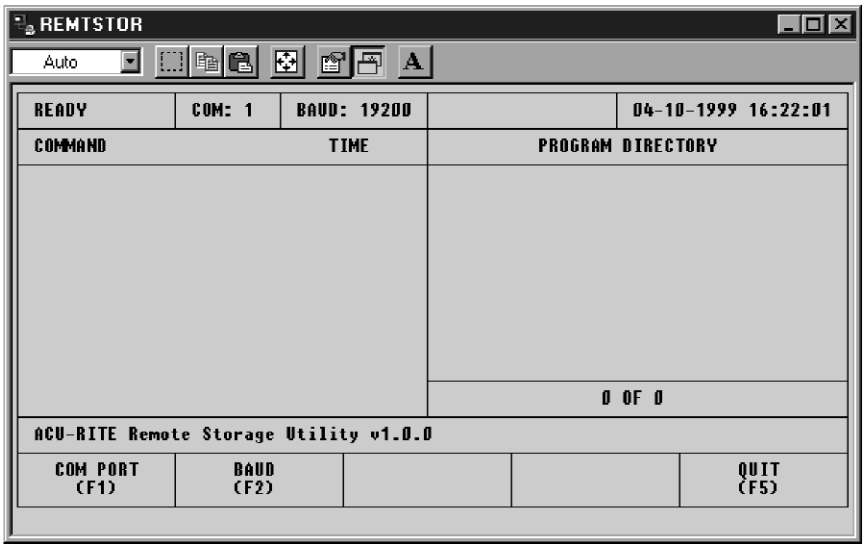
For MS DOS:

1. Power up your PC.
2. Insert the "Remote Storage" installation disk into your PC's floppy disk drive.
3. At the C:\. prompt, type **a:** then press **ENTER**.
4. After the A:> prompt, type **install**, and press **ENTER** again. This command tells the PC to copy all the files from the disk onto your PC.
5. At the next prompt, type **c:** then press **ENTER**.
6. After the C:\ prompt, type **cd remtstor** and press **ENTER**.
7. C:\REMTSTOR\> will appear. Type **remtstor** and press **ENTER**.

Your PC should now display the Remote Storage Utility screen.

Setting up Your COM Port and BAUD Rates

After the Remote Storage software has been installed on your PC, the following screen will appear:



Now you need to identify your COM (short for “communication”) port and set the BAUD rate.

- **COM Port** - COM port 1 is the program's default setting. You can switch to COM port 2 by pressing the F1 key on your PC's keyboard. Remember that the COM port setting must be available, and it must correspond with the PC port that you connected your serial cable to.

- **BAUD Rate** - Set the BAUD rate by pressing the F2 key on your PC's keyboard and scrolling through the list of BAUD rates available until the correct value appears. Check that the BAUD rate you define for your PC matches the BAUD rate you set for **MILL*PWR***.

After your COM port and BAUD rate have been set, you can begin storing part programs onto your PC using **MILL*PWR***'s serial port. (For more information about creating, saving, and deleting programs, refer to **Programming**.)

Activating the Remote Storage Feature in MILL*PWR*

Refer to **Setup** to activate the remote storage feature in your **MILL*PWR*** system.

Troubleshooting

If the communication link between **MILL*PWR*** and your PC is disrupted, an error message will appear on the **MILL*PWR*** screen. Settings that do not match and an improper serial cable connection are the most common problems and are the easiest to correct. Refer to the general **Troubleshooting Guide** section for details.

This page intentionally left blank.

Troubleshooting Guide

Introduction

This troubleshooting guide is intended to assist you with diagnosing problems should you experience any difficulties with your **MILL*PWR*** system.

Using the Table

This troubleshooting guide is arranged in three columns entitled Symptom, Probable Cause and Recommended Corrective Action. The Symptoms are listed in the order of most common, easiest to check, and least expensive to correct. First, locate the symptom that best describes the problem that needs to be corrected. Then, identify the probable cause that most closely matches the problem that you are experiencing and implement the recommended corrective action.

This troubleshooting guide pertains to the ACU-RITE **MILL*PWR*** system specifically and on occasion the machine tool itself. It does not fully address problems associated with inadequate tooling, improper feed rates and/or spindle speeds, etc.

Note: This troubleshooting guide has been written with the understanding that your ACU-RITE **MILL*PWR*** system has been properly installed by a factory-trained and certified technician.

IMPORTANT
Do not attempt to make any repairs to the operator console and/or motor assemblies on your own. Please contact your ACU-RITE distributor for repair and/or replacement assistance.

Symptom	Probable Cause	Recommended Corrective Action
Operator console displays a blank screen.	System has been idle for approximately 20 minutes and is in screen saver mode.	Check that the power indicator light, located in the upper left corner of the operator console, is illuminated. If the power indicator light is illuminated, either move the table or press any key on the operator console to restore the display. If the power indicator light is not illuminated, power has been interrupted. Follow the recommendations for "No Power."
	No power	Check that the power switch located on the rear of the operator console is on. Check that the power cord is properly connected. Check that there are no blown fuses or tripped breakers. Check that your power source meets the specifications required of the system. In the event of a power interruption, you simply have to wait until power is restored.

Symptom	Probable Cause	Recommended Corrective Action
Operator console displays a blank screen *(cont'd)*	LCD display failure	Contact your ACU-RITE distributor for repair and/or replacement procedures.
Operator console displays a flashing cursor	A floppy disk is in the floppy disk drive	Remove the floppy disk from the disk drive and cycle power to the system. Remember to use the "Find Home" routine.
	System begins to power up, but cannot get past flashing cursor	The system software has been corrupted or may not have been properly installed. Re-install the system software and power up the system again. Remember to use the "Find Home" routine.
	The floppy disk drive is not functioning properly.	Power up the system again and check that the LED (located on the front of the floppy disk drive) is illuminated. Also, listen for the spinning sound that a floppy disk drive makes when it is being accessed. Should the above recommendations not correct the problem you are experiencing, contact your ACU-RITE distributor for repair and/or replacement procedures.

Symptom	Probable Cause	Recommended Corrective Action
Operator console keypad does not function properly.	System needs to be reset.	Power down the system. Wait for approximately one minute then power the system up. Remember to reestablish machine tool zero using the "Find Home" routine. Run the Keypad Test (refer to **Diagnostics**, Section 7). Should the above recommendation not correct the problem you are experiencing, contact your ACU-RITE distributor for repair and/or replacement procedures.
System powers up, but motor(s) will not move.	Table stop button has been pressed.	Check the message bar for the following message: “Table Stop Button is Pressed.” Release the table stop button and press the **CLEAR** key.
	GRAPHICS ONLY selected in "Run Options."	Review selections in "Run Options.”
	MANUAL POSITIONING selected in "Run Options."	Review selections in "Run Options.”
	Improper Scale Connection	Check the message bar for the following message: “DSP fault-(X,Y, or Z) axis position error check failed.”

Symptom	Probable Cause	Recommended Corrective Action
System powers up, but motor(s) will not move. *(cont'd)*	Improper scale connection *(cont'd)*	Check the scale connections at the motor assemblies.
	For Z-axis Optional Linear encoder:	**For Z-axis Optional Linear encoder:**
	X-Y- and/or Z-axes precision glass scales stopped counting	While in the DRO display, move each axis manually to verify that the precision glass scales count. If the precision glass scales do not count, perform the following: Press the table stop button. Swap the X- and Y-axes servo connections. In the DRO display, move each axis manually to check if the precision glass scales count. If a precision glass scale doesn't count, follow the instructions for "The scale(s) miscount in certain areas only" below. Next, swap the X- and Z- axes servo connections and continue to check for a scale counting error. If a precision glass scale doesn't count, follow the instructions for "The scale(s) miscount in certain areas only" below. **Do not release the table stop button until you have restored the X-, Y- and Z-axes servo connectors to their proper locations!**

Symptom	Probable Cause	Recommended Corrective Action
System powers up but motor(s) will not move. *(cont'd)*	The scale(s) miscount in certain areas only	**For Z-axis Optional Linear encoder:** Remove the X- or Y-axis scale(s). Clean the glass (located inside the aluminum extrusion) and the reading head glass using a cotton swab and denatured alcohol or acetone. Re-install the scale(s) and realign the reading head(s). Do not attempt to remove the Z-axis scale. Contact your ACU-RITE distributor for repair and/or replacement. Should the above recommendation not correct the problem you are experiencing, contact your ACU-RITE distributor for repair and/or replacement procedures.
	Improper X- and/or Y-axes servo connection.	Message bar reads: "(X, Y, OR Z) SERVO UNPLUGGED FAULT." Check the X-, Y- and Z-axes servo connections located on the rear of the Operator Console.
	Motors are not receiving power	Check that the power connections for each motor assembly and that the Servo Power connector (located on the rear of the operator console) are properly seated. Enable the Motor Monitor (refer to **Diagnostics**, Section 7).

Symptom	Probable Cause	Recommended Corrective Action
Table run away	The X- and Y-axes servo connections located on the rear of the Operator Console are not connected to the correct axes.	Check that the cables are properly connected to the correct axes.
For Z-axis Optional Linear encoder:	**For Z-axis Optional Linear encoder:**	**For Z-axis Optional Linear encoder:**
Table or Quill will move in only one direction.	Position is beyond travel limits	Move within the travel limits. You may want to consider adjusting your travel limits (refer to **Diagnostics**, Section 7).
Table has stiff movement.	Table is locked.	Check the table locks and loosen as needed.
	Gibs are worn or are out of adjustment.	Refer to the machine tool operation manual for adjustment procedures.
	Not enough oil in oil reservoir or oil lines are blocked or disconnected.	Check that there is sufficient oil in the oil reservoir. Should the above recommendation not correct the problem you are experiencing, contact your ACU-RITE distributor for repair and/or replacement procedures.
System not repeating or circles are not round.	Machine tool related problems	Run the Circle Interpolation Test (refer to **Diagnostics**, Section 7). Check that the knee, ram and head bolts, turret to column bolts and head to knuckle bolts are properly fastened and secure.

Symptom	Probable Cause	Recommended Corrective Action
System not repeating or circles are not round. *(cont'd)*	Machine tool related problems *(cont'd)*	Check the gibs for wear and proper adjustment. Refer to the machine tool operation manual. Align the spindle. Lower the quill full travel with a sturdy tool inserted into the spindle. Set up a dial indicator to check movement of the spindle, then move the tool side-to-side. Total movement should be within the machine tool manufacturer's specifications between the spindle and the spindle bearings and between the quill and the bore. You may need to replace the spindle bearings. Machine tool wear. **MILLPWR** can compensate for machine tool wear through its linear error compensation (LEC) feature (refer to **Setup)**.
	Improper position feedback from precision glass scales or Z-axis Rotary encoder	Check that the mounting brackets and fasteners are tight and secure. **For Z-axis Optional Linear encoder:** Check the X- and Y-axis precision glass scale and reading head for proper installation.

Symptom	Probable Cause	Recommended Corrective Action
Incorrect dimensions	System is out of tune.	Check that the proper dimensions were entered for each step of your program. Check that the correct tool information was entered for each step of the program. Measure the tool diameter to check for tool wear. Check that the correct scale factor is being used. Perform the auto tuning procedures for all three axes. If auto tuning your system does not correct the problem, contact your ACU-RITE distributor.
Poor finish	Dull tool	Replace or sharpen the tool.
	Incorrect feed rates and/or spindle speeds	Check that the correct feed rates have been entered for each step of your program. Check your spindle speed setting.
	Gibs are worn or are out of adjustment.	Refer to the machine tool operation manual.

Symptom	Probable Cause	Recommended Corrective Action
Poor finish *(cont'd)*	Worn spindle bearings	Lower the quill full travel with a sturdy tool inserted into the spindle. Set up a dial indicator to check movement of the spindle; then move the tool side-to-side. Total movement should be within the machine tool manufacturer's specifications between the spindle and the spindle bearings; and between the quill and the bore. You may have to replace the spindle bearings.
	System is out of tune.	Perform the auto tuning procedures for both the X- and Y-axes. If auto tuning your system does not correct the problem, contact your ACU-RITE distributor.

Symptom	Probable Cause	Recommended Corrective Action
Quill handle is difficult to move.	Quill lock is engaged.	Release the quill lock.
	Inadequate lubrication	Lubricate the ballscrew with a high-quality Lithium grease.
Z-axis drive system will not engage.	The "Begin" field for the Z-axis is blank.	Check that the program step is correct and that there is information in the "Begin" field for the Z-axis.
	The Z-axis is disengaged.	RE-define FTO position. Tighten the quick release knob for the Z-axis drive system (refer to **Z-axis Conventions**).
	The Z-axis is disabled.	Enable the Z-axis (refer to **Setup**).
	The quill has reached a limit switch.	Move the quill toward the center of travel
	The drive belt is loose	Adjust the drive belt tension. Should the above recommendations not correct the problem you are experiencing, contact your ACU-RITE distributor for repair and/or replacement procedures.
Unusual chatter from the quill	The quill is over extended.	Shorten the quill distance and raise the knee.

Symptom	Probable Cause	Recommended Corrective Action
Unusual chatter from the quill *(cont'd)*	The feed rate and/or spindle speed require adjustment.	Adjust the feed rate and/or spindle speed.
	The tool is worn.	Sharpen or replace the tool.
	The tool diameter is too small for the depth of cut.	Increase tool diameter or reduce the depth of the cut.
	The tool is not secure in the tool holder.	Check that the tool is secure in the tool holder.
	The machine's ram and/or head is loose.	Tighten the nut(s) on the ram and/or head as required by the manufacturer.
	The spindle bearings are worn.	Replace the spindle bearings as instructed in the milling machine's reference manual.
Z-axis engage/disengage mechansim will not re-engage ballscrew nut or quill does not move freely.	The machine's head has been moved and is out of alignment.	Re-tram the machine's head.
		Re-tram the head and re-tighten the four (4) head coupling nuts to 50 ft/lbs. of torque.
Poor dimensional accuracy	The machine's head is out of alignment.	Re-tram the machine's head.
	The head has been moved or re-angled.	Re-tram the head and re-tighten the four (4) head coupling nuts to 50 ft/lbs. of torque.

Symptom	Probable Cause	Recommended Corrective Action
Poor dimensional accuracy *(cont'd)*	The quick release knob for the Z-axis drive is loose.	Tighten the quick release knob. Should the above recommendations not correct the problem you are experiencing, contact your ACU-RITE distributor for repair and/or replacement procedures.
BAUD rate error	**MILLPWR** and PC BAUD rate settings do not match.	Change one of the settings so that the BAUD rates for **MILLPWR** and your PC are the same.
Check sum error	**MILLPWR** and PC BAUD rate settings do not match.	Change one of the settings so that the BAUD rates for **MILLPWR** and your PC are the same.
No response from PC	BAUD rate is too slow or too fast.	Adjust the BAUD rates on both **MILLPWR** and your PC. Check that they match when you're finished adjusting them.
	MILLPWR and PC BAUD rate settings do not match	Change one of the settings so that the BAUD rates for **MILLPWR** and your PC are the same.
	Incorrect serial cable	Verify that you are using the correct type of serial cable. Refer to page 8-1, choosing a serial cable

Symptom	Probable Cause	Recommended Corrective Action
No response from PC (cont'd)	Your PC's COM port setting is different from the COM port that your serial cable is connected to.	If the COM port setting (1 or 2) on the PC screen does not correlate with the COM port that the serial cable is connected to: Change your PC's COM port setting; -OR- Disconnect the cable from the COM port that it is connected to and connect it to a different COM port.
	The selected COM port is disabled.	Change the PC's COM port setting and connect the serial cable into the appropriate port; -OR- Exit the remote storage program on your PC, then refer to your PC's operation manual for instructions on how to enable a COM port.
Time-out error	Loose serial cable	Check that the serial cable is properly connected to **MILLPWR** and to your PC.

Re-aligning (Tramming) the Spindle

Note: This does not require re-aligning the quill casting.

Note: The Z-axis is still fully functional in a tilted position.

When the head has been re-angled the following procedure should be performed so the original accuracy can be re-established. Do not loosen casting bolts or hex nuts; loosen the tramming hex stand-offs only.

You will need the following components:

- Box or open-ended wrench
- Crow's foot torque wrench 7/8"
- Dial indicator
- Dial holder

1. When the head is re-positioned to mill certain angles (see **Figure 1**) torque the four (4) coupling nuts that secure the Z-axis casting to the head to 50 ft/lbs while the head is in that position (see **Figure 2**).
2. When the milling head is repositioned to "0" degrees, use a dial indicator to fine-tune the accuracy of the head's position (see **Figure 3**).
3. Ensure the knee lock clamps on the machine saddle are tight.
4. Follow the machine manufacturer's procedure for proper milling head calibration and alignment.

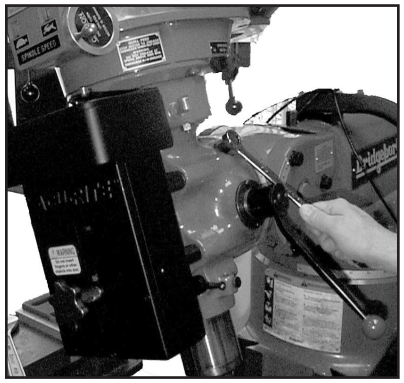


Fig. 1

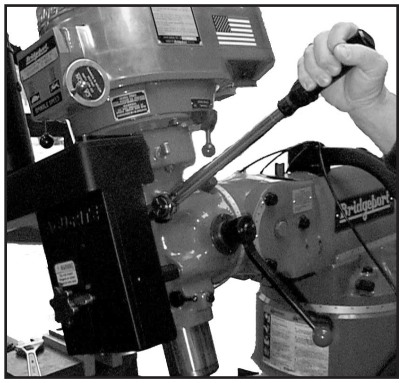


Fig. 2

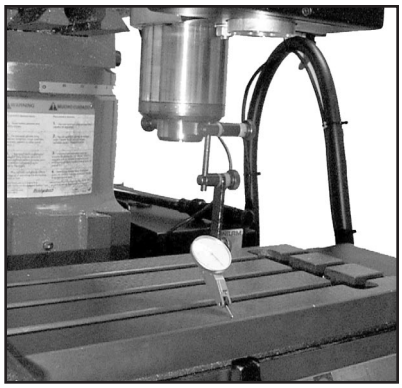


Fig. 3

5. Once the dial indicator shows a "0" reading in several locations of the table and the head is trammed in, re-torque the four (4) coupling nuts to 50 ft/lbs (see **Figure 4**). As you tighten, make sure the dial indicator maintains its "0" reading.

Note: If the 50 ft/lbs torque specification is not applied, the engage/disengage mechanism may not engage the Z-axis ballscrew nut correctly. This may also create a binding and/or undue wear to the spindle within the mill head.

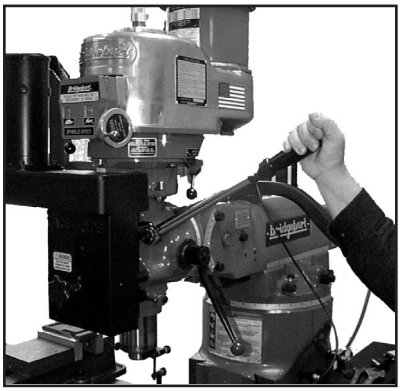


Fig. 4